Chapter 3
CFD: An Innovative and Effective Design Tool for the Food Industry

Tomás Norton and Da-Wen Sun

3.1 Introduction

Fluids are ubiquitous in nature. In fact, many if not most industrial food processes require work done on or by a fluid-based system during product development. During the design and optimization of food processes, engineers must regularly tackle problems that involve not only fluids but also complex phenomena such as heat and mass transfer, phase change, and chemical reactions. Forming a comprehensive understanding of food-processing systems is not easy because the above phenomena generally progress as invisible to the eye, and thus often their dynamics cannot be easily realized or quantified. Moreover, when presented with complex flow fields, which are governed by nonlinear dynamics, measurements alone may not be sufficient for accurate analysis, as not only can equipment be intrusive but too many measurement probes may also be required to fully diagnose a problem.

Computational fluid dynamics (CFD) is mainly concerned with the numerical solution of the partial differential equations governing transport of mass, momentum, and energy in moving fluids. However, most importantly, many processes involving the motion of fluids and heat transfer, mass transfer, and chemical reactions can be accurately described with CFD. Currently CFD is being universally used as an effective tool to investigate the spatiotemporal dynamics of many interesting processes used by the food and beverage industry such as mixing, drying, cooking, sterilization, chilling, and cold storage (Sun 2007). As a result, the application and development of CFD for food processing is now proving to play a fundamental role in problem solving for both industry and research, and its use by the scientific community has grown exponentially within the last number of years (Norton and Sun 2007). CFD has many advantages over traditional design and analysis techniques, for example:

T. Norton and D.-W. Sun (✉)
Food Refrigeration and Computerised Food Technology (FRCFT), University College Dublin, National University of Ireland, Agriculture & Food Science Centre, Belfield, Dublin 4, Ireland
e-mail: dawen.sun@ucd.ie

J.M. Aguilera et al. (eds.), *Food Engineering Interfaces*, Food Engineering Series,
DOI 10.1007/978-1-4419-7475-4_3, © Springer Science+Business Media, LLC 2011

- Because food processes can be examined in a virtual environment, designs can be achieved without encountering health and safety concerns or scaling issues associated with physical prototyping, etc.
- Many designs of a single system/process can be explored on the computer with virtual prototypes, thereby reducing wasteful trial and error techniques in industry.
- Parametric studies can be easily completed in order to come up with design solutions for a new geometry at a fraction of the cost of manufacturing the various design options.

The fundamentals of CFD applications, its general applications in the food industry, and the existing limitations and challenges that face current users of this technology have been reviewed by some authors (Xia and Sun 2002; Norton and Sun 2007). In light of this technology's rapid development over the last number of years, and the engineering challenges that face the food industry, this chapter gives a comprehensive account of the latest advances made through successful CFD applications in research. Firstly, the equations governing the physical mechanisms encountered when modeling different processes in the food industry will be discussed, followed by an overview of how they are solved in a CFD environment. An account of CFD modeling studies conducted on emerging food-processing technologies will then be highlighted. Finally, an overview will be given on the challenges still encountered in CFD modeling of food processes.

3.2 Modeling Food Processes: Solving Governing Partial Differential Equations

For all food-processing systems, partial differential equations (PDEs) can describe the intensity and spatiotemporal migration of thermodynamic and fluid-dynamic phenomena. In order to develop and optimize food-processing systems, engineers need to form a complete picture of the various mechanisms working within. Also, to avoid the expense and time associated with physical experimentation, such insight demands solutions to PDEs that describe the system. However, whether or not such solutions are achievable is dependent on the physics involved, the level of precision associated with the analysis tools at hand, and the amount of computing power available to the engineer. In fact, because PDEs are generally by their very nature time-consuming to solve, they require substantial computer resources; therefore adequate solutions have not always been attainable with prevailing computer power. In the past this meant that PDEs needed to be simplified so that they could be manageably solved by hand, and assumptions impacting the quality of a solution could easily be introduced. Furthermore, localized phenomena such as mixing, fouling, cleaning, etc. cannot be accurately determined by simple models; numerical analysis has always been required to establish the distribution of variables, that is, temperature, fluid velocity, and concentration in such systems.

Modern CFD codes have been developed using numerical algorithms that solve the nonlinear PDEs governing fluid flow, heat transfer, and many other physical phenomena. With this myriad of capabilities, CFD techniques can be used to build distributed parameter models that are spatially and temporally representative of food-processing systems, thereby allowing solutions with high levels of physical realism. The accuracy of a CFD simulation is a function of many parameters, including the level of empiricism involved, that is, via turbulence models or additional physical models; the assumptions involved, that is, Boussinesq versus the ideal gas approximation for inclusion of buoyancy effects; the simplification of both geometry and boundary conditions to reduce processing time; and whether processes modifying the physical mechanisms are included, that is, chemical kinetics. As with other modeling techniques, the greater the number of approximations, the less accurate the CFD solution will be (Verboven et al. 2004). However, if used correctly, CFD provides an understanding of the physics of a system in greater detail through nonintrusive flow, thermal, and concentration field predictions. Furthermore, after many years of development, CFD codes can now solve advanced problems related to combined convection, radiation and conduction heat transfer, flows in porous media, multiphase flows, and issues involving chemical kinetics.

Many applications of CFD in modern food processing already exist, such as for ovens, refrigerators, heat exchangers, in-place cleaning operations, spray dryers, biosensors, and pasteurization techniques, among others. CFD simulation has made giant leaps in realism with the yearly experience gained from each specialization in tandem with the continual development of commercial CFD codes. For example, in recent designs of multi-deck chilled cabinets, CFD simulations have been used to visualize the impact of various design constraints without the need for physical prototyping (Foster et al. 2005). Further simulations have revealed the impact of these refrigeration systems on the occupant comfort in retail stores. In the advancement of oven technology, CFD analyses have worked in conjunction with other mathematical models of food quality attributes to optimize variations in the weight loss and crust color of bread products prior to baking (Zhou and Therdthai 2007). Also, in modern operations like air impingement ovens, CFD has been employed so that the interaction of the jet flow pattern with the product can be understood; spatially dependent heat and mass transfer coefficients can be predicted; and equipment design parameters can be optimized (Kocer et al. 2007; Olsson and Trägårdh 2007). In recent times, three-dimensional CFD modeling of plate heat exchangers (PHEs) has allowed the milk fouling process to be simulated based on both hydrodynamic and thermodynamic principles. This fouling model permits assessing the influence of corrugation shapes and orientations on the PHE performance (Jun and Puri 2006). The current standard for CFD modeling of spray dryers has also raised the value of simulations by extending the basic models of flow patterns and particle trajectories to sub-models such as drying models, kinetic models of thermal reactions, sub-models describing product stickiness, and agglomeration models (Straatsma et al. 2007). In other areas, CFD has recently been used to model:

- Pasteurization of intact eggs (Denys et al. 2007); this application formerly had little information concerning temperatures and process times required.
- Application of biosensors in the food industry (Verboven et al. 2007a); this modeling allows the optimization of biosensor design in terms of fluid flow, mass transfer, and chemical kinetics.
- Tea fermentation and infusion (Lian 2007); CFD is used to predict the interplay between the heat transfer and enzymatic reactions during tea fermentation; optimal process conditions for desired flavor generation and functional properties can also be obtained.

All of these applications are good examples of CFD's ability to resolve problems in both novel and conventional food-processing operations, even in systems for which there is little prior knowledge. Also, since CFD can compute variables such as fluid velocity, pressure, and temperature at many thousands of locations within a virtual geometry, the solutions can be comprehensive, both spatially and temporally. The various PDEs that govern the phenomena encountered in food-processing systems are introduced next. The governing equations of fluid flow and heat transfer can be considered as mathematical formulations of the conservation laws of fluid mechanics. When applied to a fluid continuum, these conservation laws relate the rate of change of a desired fluid property to external forces. The following sections describe the governing equations for these various phenomena in Cartesian coordinates.

3.2.1 Conservation of Mass

The mathematical formulation of the law of conservation of mass is also known as the continuity equation. For a fluid element the solution for each requires a separate equation. The conservation of mass states that the net mass flowing through all the surfaces of a fluid element must be equal to the rate of accumulation in the element. For incompressible fluids, which are commonly encountered in the food industry, this means that the quantity of mass entering a fluid element must balance exactly with that leaving

$$\nabla.\vec{\mathrm{v}} = 0 \tag{3.1}$$

where $\vec{\mathrm{v}}$ consists of components of $\bar{v}_i$.

3.2.2 Conservation of Momentum

The conservation of momentum for a fluid element is also termed Newton's second law of motion, which states that the sum of the external forces acting on the fluid particle is equal to its rate of change of linear momentum:

$$\rho \frac{\partial v_i}{\partial t} + \rho \vec{v}.\nabla \vec{v}_i = -\nabla p + \mu \nabla^2 \vec{v}_i + \rho g \tag{3.2}$$

This mathematical formulation of fluid motion has been used for almost two centuries, since the emergence of three very important scientists in the field of fluid mechanics, namely, Leonhard Euler, Claude-Louis Navier, and George Gabriel Stokes. Leonhard Euler (1707–1783), a Swiss mathematician and physicist, formulated the Euler equations, which describe the motion of an inviscid fluid based on the conservation laws of fluid mechanics. The French engineer and physicist, Claude-Louis Navier (1785–1836), and Irish mathematician and physicist, George Gabriel Stokes (1819–1903), later introduced viscous transport into the Euler equations by relating the stress tensor to fluid motion. The resulting set of equations is now termed the Navier–Stokes equation for Newtonian fluids; it has formed the basis of modern-day CFD.

3.2.3 *Conservation of Energy*

The mathematical formulation of the conservation of energy is also called the first law of thermodynamics, which states that the rate of change in energy of a fluid particle is equal to the heat addition and the work done on the particle, assuming that the thermophysical properties are not temperature-dependent:

$$\rho c_p \frac{\partial \vec{v}_i}{\partial t} + \vec{v}.\nabla T = \lambda \nabla^2 T + s_T \tag{3.3}$$

As many food-processing operations involve heat transfer, and because the properties of fluids can be temperature-dependent, (3.3) is often coupled with the Navier–Stokes equation. When conjugate heat transfer is under investigation it is important to maintain continuity of thermal exchange across the fluid–solid interface, and the transport of heat in a solid structure should also be considered in CFD simulations (Verboven et al. 2004). The Fourier equation, which governs heat transfer in an isotropic solid, can be written as

$$\rho c_p \frac{\partial \vec{v}_i}{\partial t} = \lambda \nabla^2 T + s_T \tag{3.4}$$

For a conjugate heat transfer situation, where evaporation at the food surface is considered and where the heat transfer coefficient is known, the boundary condition for (3.4) may be written as follows:

$$h\left(T_{bf} - T_s\right) + \varepsilon\sigma\left(T_{bf}^4 - T_s^4\right) = -\lambda \frac{\partial T}{\partial n} \tag{3.5}$$

where ε is the emission factor coefficient and σ is the Stefan–Boltzmann constant. Heat transported by radiation and convection from air to food raises the sample temperature (Aversa et al. 2007). The solution of (3.3) can be used for food surfaces to calculate the local heat transfer coefficients as given below:

$$h = \frac{-\lambda \frac{\partial T}{\partial n}\Big|_{\text{surface}}}{\left(T_{bf} - T_s\right)} \tag{3.6}$$

Equation (3.6) can be used provided the surface temperature is assumed independent of the coefficient during calculations.

3.3 Modeling Properties of Fluids

3.3.1 Density

In many food-processing operations, variables such as temperature, concentration, and fluid velocity are functions of density variations caused by the heating and cooling of fluids. Therefore, the density of the fluid must be accurately represented in CFD computations. Since many fluid flows encountered in food-processing applications can be regarded as incompressible, there are two means of accounting for density variations, namely, the Boussinesq approximation and the ideal gas equation (Ferziger and Peric 2002). The Boussinesq approximation has been used successfully in many CFD applications (Abdul Ghani et al. 2001):

$$\rho = \rho_{\text{ref}}\left[1 - \beta\left(T - T_{\text{ref}}\right)\right] \tag{3.7}$$

The approximation assumes that the density differentials of the flow are only required in the buoyancy term of the momentum equations. In addition, a linear relationship between temperature and density, with all other extensive fluid properties being constant, is also assumed. This relationship only considers a single-component fluid medium; however, by using Taylor's expansion theorem, the density variation for a multicomponent fluid medium can also be derived.

For large temperature differences, the fluid flow becomes compressible with a strong coupling between the continuity, the momentum, and the energy equations through the equation of state; its properties (viscosity, heat conductivity) also vary with the temperature, making the Boussinesq flow approximation inappropriate and inaccurate (Ferziger and Peric 2002). Therefore, in such cases another method of achieving the coupling of the temperature and velocity fields is necessary. This can be done by expressing the density difference by means of the ideal gas equation:

$$\rho = \frac{p_{\text{ref}} M}{RT} \tag{3.8}$$

This method can model density variations in weakly compressible flows, meaning that the density of the fluid is dependent on temperature and composition but small pressure fluctuations have no influence.

3.3.2 Viscosity

Viscosity is an important fluid property that must be accurately represented in a CFD model, as it causes the resistance experienced by a fluid when flowing through any geometry. Viscosity is accurately quantified via the relationship between the shear stress in a fluid to the rate of deformation of the fluid.

Newtonian fluids are those that have a linear relationship between stress and rate of deformation, with the proportionality constant, which is actually viscosity. Any fluid that does not obey the Newtonian relationship between the shear stress and shear rate is called a non-Newtonian fluid. The shear stress in a Newtonian fluid is represented by the second term on the right-hand side of (3.1). Many food-processing media have non-Newtonian characteristics and the shear-thinning or shear-thickening behavior of these fluids greatly affects their thermal–hydraulic performance (Fernandes et al. 2006). In recent years, CFD has provided better understanding of the mixing, heating, cooling, and transport processes of non-Newtonian substances. Of the several constitutive formulas that describe the rheological behavior of substances, which include the Newtonian, power law, Bingham, and Herschel Bulkley models, the power law is the most commonly used model in food engineering applications (Welti-Chanes et al. 2005). However, there are some circumstances where modeling the viscosity can be avoided, as low velocities permit the non-Newtonian fluid to be considered Newtonian, for example, as shown by Abdul Ghani et al. (2001).

3.4 Modeling Particular Flow Regimes

3.4.1 Turbulent Flows

Turbulent flows are often encountered in the food industry owing to high flow rates and heat transfer interactions. Currently, even though the Navier–Stokes equations can be solved directly for laminar flows, it is not possible to solve the exact fluid motion in the Kolmogorov microscales associated with engineering flow regimes; thus turbulence requires modeling. For this reason, turbulence models are being developed yearly, with prediction accuracy undergoing great improvement over the last few years for a great many flow regimes. However, none of the existing turbulence models are complete; that is, their prediction performance is highly reliant on turbulent flow conditions and geometry. Without a complete turbulence model capable of predicting the average field of all turbulent flows, the present understanding of turbulence phenomena will reduce the generality of solutions. In the following sections, some of the best performing turbulence models are discussed.

3.4.1.1 Large Eddy Simulations

Large eddy simulation (LES) forms a solution given the fact that large turbulent eddies are highly anisotropic and dependent on both the mean velocity gradients and geometry of the flow domain. This is done mathematically by separating the velocity field into a resolved and sub-grid part. The resolved part of the field represents the large eddies, while the sub-grid part of the velocity represents the small scales whose effect on the resolved field is included through the sub-grid scale model. With the advent of more powerful computers, LES offers an accurate means of computing turbulent flow. However, the lengthy time involved in arriving at a solution means that it is an expensive technique, and consequently, applications of LES for CFD calculations of food processing are still uncommon (Turnbull and Thompson 2005).

3.4.1.2 Reynolds Averaged Navier–Stokes

When variables are influenced by turbulence, engineers are generally content with a statistical probability that processing variables within the flow regime (e.g., velocity, temperature, and concentration) will lie within a certain range of values. Therefore, the predictions afforded by the Reynolds-averaged Navier–Stokes equations (RANS), which determine the effect of turbulence on the mean flow field through time averaging, are in many cases sufficient. By averaging in this way, the stochastic properties of turbulent flow are essentially ignored along with six additional stresses (Reynolds stresses) emerging, which need to be modeled by a physically well-posed equation system to obtain an accurate closure of the equation system. The following paragraphs describe the common techniques used.

Reynolds Stress Models

The Reynolds stress closure model (RSM) generally consists of transport equations for the Reynolds stresses – three transport equations for the turbulent fluxes of each scalar property and one transport equation for the dissipation rate of turbulence energy. RSMs have exhibited far superior predictions for flows in confined spaces where adverse pressure gradients occur. Terms accounting for anisotropic turbulence, which are included in the transport equations for the Reynolds stresses, means that these models provide a rigorous approach to solving complex engineering flows. However, storage and execution time can be expensive for three-dimensional flows. Moreover, convergence of the RSMs has been reported to be quite poor in the literature for many flow configurations.

Turbulent Viscosity Models

The turbulent viscosity hypothesis (Boussinesq relationship) states that an increase in turbulence can be represented by an increase in the effective fluid viscosity. The Reynolds stresses are proportional to the mean velocity gradients with the effective viscosity representing the proportionality constant (Ferziger and Peric 2002). For a k–ε type turbulence model turbulence viscosity can be represented as follows:

$$\mu_t = \rho C_\mu \frac{k^2}{\varepsilon} \tag{3.9}$$

For the k–ω type turbulence without the low Reynolds number modifications, the turbulence viscosity can be represented by (Wilcox 1993):

$$\mu_t = \rho \frac{k}{\omega} \tag{3.10}$$

This hypothesis forms the foundation for many of today's most widely used turbulence models, ranging from simple models based on empirical relationships to variants of the two-equation k–ε model, which describes turbulence viscosity through turbulence production and destruction (Versteeg and Malalsekeera 1995). All turbulence viscosity models have relative merit with respect to simulating food-processing operations.

The Standard k–ε Model

The standard k–ε model (Launder and Spalding 1974), which is based on the transport equations for the turbulent kinetic energy k and its dissipation rate ε, is semi-empirical and assumes isotropic turbulence. Although it has been successful in numerous applications and is still considered an industrial standard, the standard k–ε model is limited in some respects. A major weakness of this model is that it assumes an equilibrium condition for turbulence, that is, the turbulent energy generated by the large eddies is distributed equally throughout the energy spectrum. However, energy transfer in turbulent regimes is not automatic and a considerable length of time may exist between the production and the dissipation of turbulence.

The RNG k–ε Model

The renormalization group (RNG) k–ε model (Choundhury 1993) is similar in form to the standard k–ε model, but owing to the RNG methods from which it has been analytically derived, it includes additional terms for dissipation rate development and different constants from those in the standard k–ε model. As a result, the solution accuracy of highly strained flows has been significantly improved. The calculation of the turbulent viscosity also takes into account the low Reynolds number if such a condition is encountered in a simulation. The effect of swirl on turbulence is included in the k–ε RNG model, thereby enhancing accuracy for recirculating flows.

The Realizable k–ε Model

In the realizable k–ε model (Shih et al. 1995) C_μ is expressed as a function of mean flow and turbulence properties, instead of being assumed constant, as in the case with the standard k–ε model. As a result, it satisfies certain mathematical constraints on the Reynolds stress tensor that are consistent with the physics of turbulent flows (e.g., normal Reynolds stress terms must always be positive). Also, a new model for the dissipation rate is used.

The k–ω Model

The k–ω model is based on modeled transport equations, which are solved for the turbulent kinetic energy k and the specific dissipation rate ω, that is, the dissipation rate per unit turbulent kinetic. An advantage that the k–ω model has over the k–ε model is that its performance is improved for boundary layers under adverse pressure gradients, as the model can be applied to the wall boundary, without using empirical log-law wall functions. A modification was then made to the linear constitutive equation of the k–ω model to account for the principal turbulence shear stress. This model is called the shear-stress transport (SST) k–ω model; it provides enhanced resolution of the boundary layer in viscous flows (Menter 1994).

3.4.2 Flows Containing Different Phases

Multicomponent flows occur where the species are mixed at molecular level, that is, they share the same pressure, velocity, and temperature fields. Alongside global mass continuity the transport equation for the species provides the means for updating the field of mass fractions, defining the mixture composition, via the following equation:

$$\frac{\partial e X_k}{\partial t} + \nabla \cdot (e\vec{v}_i X_k) = \nabla \cdot (D_k \nabla X_k) + S_k \tag{3.11}$$

However, as the relative size of the species becomes bigger there is deviation from the uniform flow field, and hence multiphase modeling must take over the multicomponent modeling. In multiphase flows more than one immiscible fluid is present in the flow. In CFD modeling, the term multiphase denotes a fluid system that contains fluids of different physical properties, and does not necessarily refer to the state of matter, that is, whether it is solid, liquid, or gas. In multiphase systems, multiple fluids are mixed at a macroscopic level, where the mixing scale is larger than the molecular scale. In general, two-phase fluid systems are encountered in the food industry; however, the same concepts for their modeling apply to systems having many more phases. Multiphase flows can be broadly placed into the following categories:

1. Continuous; all the fluids involved in the flow are continuous such as liquid–liquid or liquid–gas flows where a free and distinct interface exists between fluids, for example, bubbly fluids or immiscible fluids.
2. Continuous–dispersed; namely, liquid–solid flows that contain a dispersed phase within a primary continuous phase, for example, particle-laden flows such as chunky soups, spray dryers, and fluidized beds.

In multiphase flows, each phase competes for the same volume, resulting in interactions between phases due to their proximity. Owing to the abrupt difference between the properties of the various phases, mass, momentum, and energy are exchanged between the phases. Such exchanges need to be accounted for in a CFD model, and in many cases these interactions are complex and specific to the type of flow being modeled.

Multiphase modeling has to provide models for tracking phases and/or predicting their distribution in space and time. Three approaches are generally used: the volume of fluid model (VOF), Eulerian–Eulerian model, and Lagrangian-Eulerian model, and are briefly discussed in the following sections.

3.4.2.1 Volume of Fluid Model

This simple multiphase model is well suited to simulate flows of immiscible fluids on numerical meshes that are capable of solving the interface between the mixture's phases. In other words, the VOF is a volume fraction tracking technique, which is highly effective when the shape of the interface between phases is important. Moreover, by using the VOF technique no additional modeling of interphase interactions is required, as all phases are assumed to share the same velocity, pressure, and temperature fields. The VOF model can calculate large-scale breakups and agglomerations such as sloshing effects caused by a moving water tank. If the breakup is too small to calculate, that is, during the formation of air bubbles in water or water droplets in air, then very high mesh refinement is required for accurate predictions.

3.4.2.2 Eulerian–Eulerian Model

Models simulating the distribution of phases via the Eulerian–Eulerian concept look at the presence and transport of phases in space, rather than tracking individual particles of phases. In the Eulerian approach, the phases are treated as interacting and interpenetrating continua (Nijdam et al. 2006). The phases share the same volume, penetrate each other in space, and exchange mass, momentum, and energy with each other. Each phase is described by its distinctive physical properties and has its own velocity, pressure, concentration, and temperature field. Coupling of the phases is achieved through pressure and interphase exchange coefficients.

The Eulerian–Eulerian model is applicable for continuous–dispersed as well as continuous–continuous systems. For continuous–dispersed systems, the velocity of each phase is computed using the Navier–Stokes equations. The dispersed phase may be in the form of particles, drops, or bubbles. The forces acting on the dispersed phase are modeled using empirical correlations and included as part of the interphase transfer terms. Drag, lift, gravity, buoyancy, and virtual mass effects are some of the forces that may be acting on the dispersed phase. These forces are computed for an individual particle and then scaled by the local volume fraction to account for multiple particles.

3.4.2.3 Lagrangian–Eulerian Model

This model is applicable to modeling continuous–dispersed systems and is very often referred to as a discrete particle model or particle transport model. The primary phase is continuous, whereas the secondary phase is discrete and may be composed of particles, drops, or bubbles. The continuous phase flow is computed by solving the Navier–Stokes equations, and using their solutions in the flow trajectories, as well as the heat and mass transfer to and from the discrete phase, which is computed by solving the appropriate ordinary differential equations. The dispersed phase is represented by tracking a small number of representative particle streams; coupling between the continuous and discrete phases is achieved by including appropriate interaction terms in the equation set. Lagrangian–Eulerian modeling works best when the motion of particles is dominated by their interaction with the continuous phase rather than with each other, that is, when the dispersed phases are dilute. Moreover, this model is mainly applicable for low volume fractions of the dispersed phase, as the volume displacement caused by the discrete phase is not taken into account.

3.4.3 Modeling Flows through Porous Media

Many large-scale processes in the food industry may have the potential to be grid point demanding in CFD models owing to the complex geometry of the modeled structures. For example, to predict the detailed transfer processes within a cold store containing stacked foods one must mesh all associated geometry with a complex unstructured or body-fitted system, which is a highly arduous, and in many cases, inaccessible task. In any case, both computational power and CFD algorithms have not yet reached such levels of maturity wherein these types of computations can be achieved. Therefore, other methods must be used to exploit the physical relationships that exist on a macroscopic level and sufficiently represent the dynamic flow effects representative of the modeled material. The porous media assumption relates the effects of particle size and shape, alignment with airflow, and void fraction to the pressure drop over the modeled products. This method basically applies Darcy's law to a porous media by relating the velocity drop through the pores to the pressure drop over the material. An extension of this law to account for most commonly encountered nonlinear relationships between pressure drop and velocity is represented by the Darcy–Forcheimer equation:

$$\frac{\partial p}{\partial x} = -\frac{\mu}{K} v + \rho C_2 v^2 \tag{3.12}$$

Equation (3.12) represents the most common relationship used to describe pressure drop across packed beds. In the CFD model this equation is added as an additional sink term to the momentum equation. The general relationships employed to determine both the permeability and inertial loss coefficient can be obtained by inference from the Ergun equation. However, considerable information regarding the detailed flow and transfer processes taking place within the stacked material is lost in this type of modeling strategy. Therefore, before modeling a porous media one must ensure that the parameters in the momentum source terms represent the physical media as closely as possible.

3.5 Numerical Methods Used by CFD Code Developers

CFD code developers have a choice of many different numerical techniques to discretize the transport equations. The most important of these include finite difference, finite elements, and finite volume. The finite difference technique is the oldest one used, and many examples of its application in the food industry exist. However, due to difficulties in coping with irregular geometry, finite difference is not commercially implemented. Furthermore, the current trend of commercial CFD coding is aimed toward developing unstructured meshing technology capable of handling the complex three-dimensional geometries encountered in industry. Therefore, the prospects of finite difference being used in industrial CFD applications seem limited.

Finite element methods have historically been used in structural analysis where the equilibrium of the solution must be satisfied at the node of each element. Nicolaï et al. (2001) provided a short introduction on using the finite elements method in conduction heat transfer modeling, but it will not be discussed here. It is sufficient to say that as a result of the weighting functions used in this method, obtaining a three-dimensional CFD solution with a large number of cells is impractical at present. Therefore, finite elements are not generally used by commercial CFD developers, especially since many of these CFD codes are marketed toward solving aerodynamic problems. Nonetheless, finite elements methods have been used in the modeling of electromagnetic heating in microwave ovens (Verboven et al. 2007b); vacuum microwave drying (Ressing et al. 2007); radio frequency heating of food; and conduction and mass transport during drying (Aversa et al. 2007). Therefore, it would seem the finite elements method is amenable to the modeling of novel thermal processes if the details of fluid flow do not need explicit quantification.

With finite volume techniques the integral transport equations governing the physical process are expressed in conservation form (divergence of fluxes); the volume integrals are then converted to surface integrals using Gauss's divergence theorem. This is a direct extension of the control volume analysis that many engineers use in thermodynamics and heat transfer applications, etc., so it can be easily interpreted. Thus, expressing the equation system through finite volumes forms a physically intuitive method of achieving a systematic account of the changes in mass, momentum, and energy, as fluid crosses the boundaries of discrete spatial volumes within the computational domain. Also, finite volume techniques yield algebraic equations that promote solver robustness, adding further reasons to why many commercial developers implement this technique.

3.6 Applications of CFD in Food Industry

3.6.1 Sterilization

3.6.1.1 Canned Foods

Sterilization is a conventional thermal process that can be modeled with CFD. In this process rapid and uniform heating is desirable to achieve a predetermined level of sterility with minimum destruction in the color, texture, and nutrients of food products (Tattiyakul et al. 2001). Siriwattanayotin et al. (2006) used CFD to investigate sterilization value (F_0) calculation methods, and concluded that when F_0 was determined using the "Thermal Death Time" (TDT) approach, the process time to achieve the desired temperature in a sugar solution was underestimated if the surrounding temperature was lower than the reference value.

Canned viscous liquid foods such as soup, carboxyl-methyl cellulose (CMC), or corn starch undergoing sterilization have been simulated with CFD. In most cases where natural convection occurs, fluid velocities and shear rates are rather low and thus non-Newtonian fluids can be assumed as Newtonian (Abdul Ghani et al. 2003). Neglecting heat generation due to viscous dissipation is another assumption that is generally made. In such simulations, CFD has shown the transient nature of the slowest heated zone (SHZ), and has illustrated the large amount of time needed for heat to be transferred throughout food, as well as the sharp heterogeneity in the temperature profile when the process is static.

CFD has also been used to study the effect of container shape on the efficiency of the sterilization process (Varma and Kannan 2005, 2006). Conical vessels pointing upward were found to reach appropriate sterilization temperature the quickest (Varma and Kannan 2005). Full cylindrical geometries performed best when sterilized in a horizontal position (Varma and Kannan 2006).

3.6.1.2 Pouched Foods

In recent years, CFD has provided a rigorous analysis of the sterilization of three-dimensional pouches containing liquid foods (Abdul Ghani et al. 2002; Abdul Ghani and Farid 2006). Coupling first-order bacteria and vitamin inactivation models with the fluid flow has allowed prediction of the transient temperature, velocity, and concentration profiles of both the bacteria and ascorbic acid during natural convection. The concentrations of bacteria and ascorbic acid after heat treatment of pouches filled with the liquid food were measured, and close agreement was found with the numerical predictions. The SHZ was found to migrate during sterilization until eventually resting in a position at a distance about 30% from the top of the pouch. As expected, the bacterial and ascorbic acid destruction was seen to depend on both the temperature distribution and flow pattern.

3.6.2 Pasteurization

CFD has been used to predict the transient temperature and velocity profiles during pasteurization of intact eggs (Denys et al. 2007). Owing to its ability to account for complex geometries, heterogeneous initial temperature distributions, transient boundary conditions, and nonlinear thermophysical properties, CFD has permitted a comprehensive understanding of this thermal process. Such analysis has allowed the gap in knowledge of this area to be filled, because up to recently little information was available on the correct processing temperatures and times for safe pasteurization, without loss of functional properties. In the series of studies published on this topic by Denys et al. (2007), a procedure to determine the surface heat transfer coefficient using CFD simulations of eggs filled with a conductive material of known thermal properties was first developed, after which conductive and convective heating processes in the egg were modeled (Denys et al. 2007). This revealed that, similar to the phenomena noted by Abdul Ghani et al. (1999) for canned foods, the cold spot moved during the process toward the bottom of the egg. The location of the cold zone in the yolk was predicted to lie below its geometrical center, even when the yolk was positioned at the top of the egg. It was concluded that no convective heating takes place in the egg yolk during processing.

3.6.3 Aseptic Processing

3.6.3.1 Plate Heat Exchangers for Milk Processing

In the dairy industry it is essential to heat-treat milk products in a continuous process, as on the one hand, it is necessary to promote microbial safety and increase the shelf life of milk. However, on the other hand, efficient plant processes are also desirable, but the adverse influences of heat on the sensory and nutritional properties of the final milk product act as a hindrance to efficient thermal processing.

Many CFD studies of PHEs exist, and have presented different techniques for geometry optimization, for example, corrugation shape or the optimization of other process parameters such as inlet and outlet positions and PHE–product temperature differences (Grijspeerdt et al. 2007; Jun and Puri 2005). Grijspeerdt et al. (2007) investigated the effect of large temperature differences between the product and PHE, and noted that the larger the difference, the greater is the opportunity for fouling. While fouling has been studied by the same group of authors, Jun and Puri (2005) were the first to couple a fouling model with a three-dimensional thermal–hydraulic model with CFD simulations. In this study, Jun and Puri investigated the influence of various PHE designs on fouling rates, comparing those typically used in the dairy industry with those used in the automobile industry.

3.6.3.2 Plate Heat Exchangers for Yoghurt Processing

Fernandes et al. (2005, 2006) studied the cooling of stirred yoghurt in PHEs with CFD simulations in order to investigate the thermal–hydraulic phenomena. They modeled the rheological behavior of yoghurt via a Herschel–Bulkley model. In addition to accounting for this rheological behavior, they also provided a high level of precision in the PHE geometrical design and the imposed boundary conditions. During the course of these studies, it was found that due to the higher Prandtl numbers and shear thinning effects of the yoghurt, the Nusselt numbers of the fully developed flows were more than ten times higher than those of water. This result presented a substantial thermal–hydraulic performance enhancement in comparison with that from Newtonian fluids. Furthermore, it was shown that PHEs with high corrugation angles may provide better opportunities for the gel structure breakdown desired during the production stage of stirred yoghurt.

3.6.4 Drying

3.6.4.1 Fluidized Beds

Because of the complex interactions that occur during fluidized bed drying, empirical correlations are only valid for a certain range of conditions, and CFD simulations have been the only means of providing accurate information on the flow phenomena. However, similar to other drying applications, difficulties exist in modeling the interactions between the solid and liquid phases, as well as limitations in computing power, and consequently only a limited number of CFD simulations exist.

The Eulerian–Eulerian approach offers the most efficient way of representing the two phases in this type of system, considering the present level of computer power available and the large number of granules (grain) in a typical system; consequentially its use has been preferred over discrete methods by some researchers. Szafran and Kmiec (2004), via the multi-fluid granular kinetic model of Gidaspow et al. (1992) and the k–ε model, used this approach in their CFD simulations of the transport mechanisms in the spouted bed.

A major difficulty encountered in modeling spouted bed dryers has been the excessive computing times required to simulate only a fraction of the drying process. For example, owing to the small time steps required to resolve the instabilities in the flow regime, it would take a two-dimensional CFD simulation almost 1 year to simulate 1 h of drying (Szafran and Kmiec 2004). Thus, at present CFD can only be used as a tool that permits a deeper understanding of the flow patterns and their effects on the drying kinetics rather than for design and optimization.

3.6.4.2 Spray Drying

Spray drying is another traditional drying technique and is used to derive powders from products, with its main objective being to create a product that is easy to store, handle, and transport. CFD has been a necessary requisite for accurate spray dryer modeling, and has been employed for over 10 years now (Langrish and Fletcher 2003).

One of the big difficulties when using CFD software packages in spray dryer modeling is that owing to the presence of both solid and fluid, the mass transport limitations within a droplet cannot be easily taken into account, and therefore sub-models must be included to do so; for accurate solutions these must be used alongside many other sub-models that account for other phenomenological aspects (Straatsma et al. 2007). In a recent study by Straatsma et al. (2007), sub-models for mass transport, inter-particle collision, agglomeration, thermal reactions, and stickiness were implemented with an Lagrangian-Eulerian model of an industrial dryer. The CFD simulations allowed the authors (Straatsma et al. 2007) to assess the agglomeration size of the particles and the stickiness of the particles colliding with each other and the wall, and as a consequence allowed the fouling liability of the dryer to be evaluated.

3.6.4.3 Forced Convection Drying

Because of the complex geometry usually encountered in drying applications, theoretical studies are often not applicable; to obtain the spatial distribution of transfer coefficients with reasonable accuracy it is necessary to solve the

Navier–Stokes equations in the product surroundings. From the distribution of transfer coefficients the correct temperature and moisture profiles within the product can be predicted, so that the drying process can be optimized. Kaya et al. (2007) used this approach with CFD to determine the transfer coefficients; then the heat and mass transfer within the food was simulated with an external program. However, it is also possible to do this within the CFD package by determining the heat transfer coefficient, and the mass transfer with the Lewis relation, once a transient solution is performed.

3.6.5 Cooking

3.6.5.1 Natural Convection Ovens

Electric ovens are commonly used household appliances that rely on conjugate thermal exchange to produce the desired cooking effect in a foodstuff. For that reason, CFD is an appropriate tool to quantify the internal thermal field. A thorough investigation into the thermal profile of an electrical oven, operated under both broil and bake modes, was completed by Mistry et al. (2006). The solution first obtained from the steady-state analysis yielded a flow field, which opposed that evident from experimental observation. This was addressed by imposing an artificial, that is, a "numerical," vent suction pressure, the value of which was tweaked until thermal field predictions corresponded with experimental measurements. Full cycling times, employing intermittent ON/OFF operation of heaters, were also simulated for both the broil and bake cycles. From the comparison of predictions, the broil cycle was confirmed to be less efficient, with a notable heterogeneity in temperature profile, owing to temperature stratification; this underscored the fact that the main thermal exchange in this cycle was due to radiation.

3.6.5.2 Forced Convection Ovens

Application of CFD in jet impingement oven systems provides detailed understanding of the effect of different oven geometries as well as object geometries on the system performance. A full three-dimensional CFD model of a multiple jet impingement oven was developed by Kocer et al. (2007). Convection and conduction heat transfers were coupled. However, as the thermal exchange was a result of jet impingement only, radiation was ignored with a small compromise in accuracy. Moisture transfer was not considered. Kocer et al. (2007) then determined a correlation for the average Nusselt number in terms of Reynolds number for multiple jets impinging on the surface of a cylindrical model cookie, which indicated the strong dependence of surface heat transfer coefficient on velocity of the jet.

3.6.5.3 Baking Ovens

Recent CFD modeling studies of the bread-baking process have looked at the two-dimensional physical representation, coupling convection, and radiative heat transfer via the discrete ordinate (DO) model (Wong et al. 2007a). Moreover, the density, heat capacity, and thermal conductivity were allowed to vary with temperature. However, some discrepancies between predictions and measurements of the actual baking process were found, especially for those comparisons made at the dough center, which were probably caused by no modeling of the moisture transport in the dough and evaporation kinetics. Moreover, the confining effect afforded by the two-dimensional model was seen to cause lack of correspondence in the validation study.

3.7 Challenges in Use of CFD in the Food Industry

3.7.1 Improving the Efficiency of the Solution Process

Insight into the numerical abilities of CFD packages is important if one needs to solve the problems of excessive computing times. Taking the parallelization features of commercial CFD codes as an example, these can allow a solution to be formed quicker, via domain decomposition, as long as the computing power is available and Lagrangian particle tracking is not employed. Alongside this, the solving techniques employed in commercial CFD codes have also been found to play a major role in efficiency. Fletcher et al. (2006) noted how segregated solvers and coupled solvers can bring different attributes to solution progression, and found that owing to the reduced levels of “random noise” introduced, the coupled solver permitted a high level of control over the solution process, allowing efficient and accurate predictions of the transient evolution of the flow instability in a spray dryer, when compared to the segregated solver.

Simplifying the geometrical representation of CFD models can also cut down on both pre-processing and solving time. The two-dimensional modeling technique assumes that the length of a system is much greater than its other two dimensions, and that the process flow is normal up to this length. As the effects of the confining geometry are essentially disregarded, accurate judgment of whether the process is amenable to the two-dimensional assumption is required.

3.7.2 CFD to Control Food Processes

All processes in the food industry are performed under controlled conditions. Unfortunately, due to the nonlinearity of the transport phenomena, CFD techniques are not yet amenable to the online control of thermal processes; reduced order models, which use statistical data to manipulate the process variables via controlled inputs, are more appropriate. However, this does not mean that the actions of a control system cannot be modeled by CFD. Wong et al. (2007b) were the first to implement a control system within a CFD model to simulate its performance. Such abilities undoubtedly provide benefits during the pre-design or optimization stages of system development.

3.7.3 Turbulence

One of the main issues facing the food industry over the last two decades is the fact that most turbulence models have been shown to be application-specific. At the present time, there are many turbulence models available; however, until a complete turbulence model capable of predicting the average field of all turbulent flows is developed the CFD optimization of many thermal processes will be hampered. The reason is that in every application many different turbulence models must be applied until the one that gives the best predictions is found. The closest to the complete turbulence model thus far is the LES, which uses the instantaneous Navier–Stokes equations to model large-scale eddies, with smaller scales solved with a sub-grid model. However, using the LES model demands large amounts of computer resources, which may not be presently achievable.

3.7.4 Need for Sensitivity Analysis

In CFD simulations, the boundary conditions must be adequately matched to the physical parameters of the process, with the precision of similarity being conditioned by the mechanism under study and the level of accuracy required. Even when this is done, the CFD solution still may not be a correct physical representation of the physical system. This was shown by Mistry et al. (2006), who found that an artificial pressure differential was required to predict the correct flow patterns in an oven heated by natural convection. Such results suggest the importance of sensitivity analysis studies alongside experimental measurements in the early stages of model development. Sensitivity analyses are also necessary for turbulence model specification, or turbulence model tuning via inlet conditions, and for CFD model simplification.

3.8 Conclusions

CFD has played an active part in the design of food operation processes for over a decade now. In recent years simulations have reached higher levels of sophistication, as application-specific models can be incorporated into the software with ease via user-defined files. The importance of maintaining a high level of accuracy via circumspect choices made during model development is evident from the reviewed studies, as many of these studies provide detailed validation exercises. Undoubtedly, with current computing power progressing unrelentingly, it is conceivable that CFD will continue to provide explanations for transport phenomena, leading to better design of processes in the food industry.

3.9 Nomenclature

C_μ	Empirical turbulence model constant
c_p	Specific heat capacity (W kg^{-1} K^{-1})
D	Diffusion coefficient
E	Mole fraction
g	Acceleration due to gravity (m s^{-2})
K	von Karman constant
M	Molecular weight (kg $kmol^{-1}$)
Nu	Nusselt number
p	Pressure (Pa)
R	Gas constant (J $kmol^{-1}$ K^{-1})
Re	Reynolds number
K	Turbulent kinetic energy (m^2 s^{-2})
s_T	Thermal sink or source (W m^{-3})
s_c	Concentration sink or source (mol m^{-3})
T	Temperature (K)
Tu	Turbulence intensity (%)
t	Time (s)
U	Velocity component (m s^{-1})
$\vec{v}_i$	Velocity component (m s^{-1})
x	Cartesian coordinates (m)
X_k	Volume fraction of k

3.9.1 Greek Letters

ρ	Density (kg m^{-3})
μ	Dynamic viscosity (kg m^{-1} s^{-1})
β	Thermal expansion coefficient (K^{-1})
λ	Thermal conductivity (W m^{-1} K^{-1})
ε	Turbulent dissipation rate (m^2 s^{-3})
φ	Transported quantity
Γ	Diffusion coefficient of transported variable

ϑ Diffusivity of the mass component in the fluid (m^2 s^{-1})
ω Specific dissipation (s^{-1})
μ_t Turbulent viscosity (kg m^{-1} s^{-1})

3.9.2 Subscripts

i Cartesian coordinate index
bf Bulk fluid
s Surface
V Mesh element volume
A Area of mesh element
m Mean
t Turbulent
k *k*th phase

References

Abdul Ghani AG, Farid MM (2006) Using the computational fluid dynamics to analyze the thermal sterilization of solid–liquid food mixture in cans. Innovative Food Science & Emerging Technologies 7(1–2):55–61
Abdul Ghani AG, Farid MM, Chen XD (2002) Theoretical and experimental investigation of the thermal inactivation of *Bacillus stearothermophilus* in food pouches. J Food Eng 51(3):221–228
Abdul Ghani AG, Farid MM, Chen XD, Richards P (1999) An investigation of deactivation of bacteria in a canned liquid food during sterilization using computational fluid dynamics (CFD). J Food Eng 42:207–214
Abdul Ghani AG, Farid MM, Chen XD, Richards P (2001) Thermal sterilization of canned food in a 3-D pouch using computational fluid dynamics. J Food Eng 48:147–156
Abdul Ghani AG, Farid MM, Zarrouk SJ (2003) The effect of can rotation on sterilization of liquid food using computational fluid dynamics. J Food Eng 57:9–16
Aversa M, Curcio S, Calabro V, Iorio G (2007) An analysis of the transport phenomena occurring during food drying process. J Food Eng 78(3):922–932
Choundhury D (1993) Introduction to the renormalization group method and turbulence modelling. Fluent Inc. Technical Memorandum TM-107
Denys S, Pieters J, Dewettinck K (2007) CFD analysis of thermal processing of eggs, chapter 14. In: Da-Wen Sun (ed) Computational fluid dynamics in food processing. CRC Press, Boca Raton, FL, pp 347–381
Fernandes CS, Dias R, Nóbrega JM, Afonso IM, Melo LF, Maia JM (2005) Simulation of stirred yoghurt processing in plate heat exchangers. J Food Eng 69:281–290
Fernandes CS, Dias RP, Nobrega JM, Afonso IM, Melo LF, Maia JM (2006) Thermal behaviour of stirred yoghurt during cooling in plate heat exchangers. J Food Eng 69:281–290
Ferziger JH, Peric M (2002) Computational methods for fluid dynamics. Springer-Verlag, Berlin/Heidleberg, pp 1–100
Fletcher DF, Guo B, Harvie DJE, Langrish TAG, Nijdam JJ, Williams J (2006) What is important in the simulation of spray dryer performance and how do current CFD models perform? Appl Math Model 30(11):1281–1292

Foster AM, Madge M, Evans JA (2005) The use of CFD to improve the performance of a chilled multideck retail display cabinet. Int J Refrig 28:698–705

Gidaspow D, Bezburuah R, Ding J (1992) Hydrodynamics of circulating fluidized beds, kinetic theory approach. Fluidization VII, Proceedings of the 7th Engineering foundation conference on fluidization, Brisbane, Australia, pp 75–82

Grijspeerdt K, Vucinic D, Lacor C (2007) CFD modelling of the hydrodynamics of plate heat exchangers for milk processing, chapter 19. In: Da-Wen Sun (ed) Computational fluid dynamics in food processing. CRC Press, Boca Raton, FL, pp 403–417

Jun S, Puri VM (2005) 3D milk-fouling model of plate heat exchangers using computational fluid dynamics. Int J Dairy Technol 58(4):214–224

Jun S, Puri VM (2006) A 2D dynamic model for fouling performance of plate heat exchangers. J Food Eng 75(3):364–374

Kaya A, Aydin O, Dincer I (2007) Numerical modeling of forced-convection drying of cylindrical moist objects. Numer Heat Transf A-Appl 51(9):843–854

Kocer D, Nitin N, Karwe MV (2007) Applications of CFD in Jet impingement oven, chapter 19. In: Da-Wen Sun (ed) Computational fluid dynamics in food processing. CRC Press, Boca Raton, FL, pp 469–487

Langrish TAG, Fletcher DF (2003) Prospects for the modelling and design of spray dryers in the 21st century. Drying Technol 21(2):197–215

Launder BE, Spalding DB (1974) The numerical computation of turbulent flows. Comput Meth Appl Mech Eng 3:269–289

Lian G (2007) CFD simulation of multi-physical multi-(bio) chemical interactions of tea fermentation and infusion, chapter 24. In: Da-Wen Sun (ed) Computational fluid dynamics in food processing. CRC Press, Boca Raton, FL, pp 478–505

Menter FR (1994) Two-equation eddy-viscosity turbulence models for engineering applications. AIAA J 32(8):1598–1604

Mistry H, Ganapathi-subbu, Dey S, Bishnoi P, Castillo JL (2006) Modeling of transient natural convection heat transfer in electric ovens. Appl Therm Eng 26(17–18):2448–2456

Nicolaï BM, Verboven P, Scheerlinck N (2001) Modelling and simulation of thermal processes, Chapter 6. In: Philip Richardson (ed) Thermal technologies in food processing. Woodhead Publishing, Cambridge/England, pp 91–109

Nijdam JJ, Guo BY, Fletcher DF, Langrish TAG (2006) Lagrangian and Eulerian models for simulating turbulent dispersion and coalescence of droplets within a spray. Appl Math Model 30(11):1196–1211

Norton T, Sun D-W (2007) An overview of CFD applications in the food industry, chapter 1. In: Da-Wen Sun (ed) Computational fluid dynamics in food processing. CRC Press, Boca Raton, FL, pp 1–43

Olsson E, Trägårdh C (2007) CFD modelling of jet impingement during heating and cooling of foods, chapter 20. In: Da-Wen Sun (ed) Computational fluid dynamics in food processing. CRC Press, Boca Raton, FL, pp 478–505

Ressing H, Ressing M, Durance T (2007) Modelling the mechanisms of dough puffing during vacuum microwave drying using the finite element method. J Food Eng 82(4):498–508

Shih TH, Liou WW, Shabbir A, Zhu J (1995) A new k–ε eddy viscosity model for high Reynolds number turbulent flows—model development and validation. Comput Fluids 24(3):227–238

Siriwattanayotin S, Yoovidhya T, Meepadung T, Ruenglertpanyakul W (2006) Simulation of sterilization of canned liquid food using sucrose degradation as an indicator. J Food Eng 73:307–312

Straatsma J, Verdurmen REM, Verscheuren M, Gunsing M, de Jong P (2007) CFD simulation of spray drying of food products, Chapter 10. In: Da-Wen Sun (ed) Computational fluid dynamics in food processing. CRC Press, Boca Raton, FL, pp 249–287

Sun D-W (ed) (2007) Computational fluid dynamics in food processing. CRC Press, Boca Raton, FL

Szafran RG, Kmiec A (2004) CFD modeling of heat and mass transfer in a spouted bed dryer. Ind Eng Chem Res 43(4):1113–1124

Tattiyakul J, Rao MA, Datta AK (2001) Simulation of heat transfer to a canned corn starch dispersion subjected to axial rotation. Chem Eng Process 40:391–399

Turnbull J, Thompson CP (2005) Transient averaging to combine large eddy simulation with Reynolds averaged Navier-Stokes simulations. Comput Chem Eng 29:379–392

Varma MN, Kannan A (2005) Enhanced food sterilization through inclination of the container walls and geometry modifications. Int J Heat Mass Transfer 48:3753–3762

Varma MN, Kannan A (2006) CFD studies on natural convective heating of canned food in conical and cylindrical containers. J Food Eng 77(4):1027–1036

Verboven P, Atalay YT, Vermeir SE, Nicolaï BM, Lammertyn J (2007a) CFD design and optimization of biosensors for the food industry, chapter 26. In: Da-Wen Sun (ed) Computational fluid dynamics in food processing. CRC Press, Boca Raton, FL, pp 478–505

Verboven P, de Baerdemaeker J, Nicolaï BM (2004) Using computational fluid dynamics to optimise thermal processes, chapter 4. In: Philip Richardson (ed) Improving the thermal processing of foods. Woodhead Publishing, Cambridge/England, pp 82–102

Verboven P, Datta AK, Nicolaï BM (2007b) Computation of airflow effects in microwave and combination heating, Chapter 12. In: Da-Wen Sun (ed) Computational fluid dynamics in food processing. CRC Press, Boca Raton, FL, pp 313–331

Versteeg HK, Malalsekeera W (1995) An introduction to computational fluid dynamics. Longman, Harlow, UK, pp 1–100

Welti-Chanes J, Vergara-Balderas F, Bermúdez-Aguirre D (2005) Transport phenomena in food engineering: basic concepts and advances. J Food Eng 67:113–128

Wilcox DC (1993) Comparison of 2-equation turbulence models for boundary-layer flows with pressure gradient. AIAA J 31(8):1414–1421

Wong SY, Zhou WB, Hua JS (2007a) CFD modeling of an industrial continuous bread-baking process involving U-movement. J Food Eng 78(3):888–896

Wong SY, Zhou WB, Hua JS (2007b) Designing process controller for a continuous bread baking process based on CFD modelling. J Food Eng 81(3):523–534

Xia B, Sun D-W (2002) Applications of computational fluid dynamics (CFD) in the food industry: a review. Comput Electron Agric 34:5–24

Zhou W, Therdthai N (2007) Three-dimensional CFD modelling of a continuous baking process, chapter 11. In: Da-Wen Sun (ed) Computational fluid dynamics in food processing. CRC Press, Boca Raton, FL, pp 287–213