The application of computational fluid dynamics in the food industry

Gordon Scott and Philip Richardson

Computational fluid dynamics (CFD) is a numerical technique for the solution of the equations governing the flow of fluids inside a defined flow geometry. CFD has seen applications in many different processing industries, but it is only in recent years that the technique has been applied to food processing applications, including air flow in clean rooms, ovens and chillers; flow of foods in continuous-flow systems; and convection patterns during in-container thermal processing. In this review article, we discuss the current state of play of CFD in the food industry and also the future advances in the technique that will guarantee its use as a tool for food process engineers.

CFD is a simulation tool for the solution of fluid flow and heat transfer problems. It comprises the Navier–Stokes transport equations, describing the conservation of mass, momentum and energy, which are solved numerically to give predictions of, for example, velocity, shear, temperature and pressure profiles inside the system being studied. The origins of CFD can be found in the automotive, aerospace and nuclear industries, and it is a technique that has a variety of applications in different processing industries. However, it is only in recent years that CFD has been applied to food processing. This article reports on some of the applications of CFD found in the food industry in recent years and aims to illustrate the benefits that can be achieved. The separate components of a generic CFD package are discussed along with some of the underlying physics that are solved. Finally, a discussion of what future developments need to be made to ensure the continued use of CFD in the food industry is given.

CFD and the food industry

CFD has only recently been applied to food processing applications. Several reviews have been written on the general application of CFD to transport processes[1] and to food processing in particular[2,3]. These reviews have listed clean-room design, refrigerated transport[4], static mixers[5] and pipe flow[5,6] as suitable for analysis by CFD. This section reviews some further reported applications of CFD in the food industry and aims to highlight what benefits can be gained by the food processor or engineer through using CFD.

The development of CFD software codes in other processing sectors, such as the aerospace and automotive industries, has resulted in CFD being used primarily as a tool for predicting air flow movement over, for example, planes and cars. Environmental studies have used CFD for the prediction of air flow around buildings. It is therefore appropriate to use CFD for the prediction of air flow movement inside items of food processing equipment.

Gordon Scott and **Philip Richardson** (corresponding author) are at the Campden & Chorleywood Food Research Association, Chipping Campden, UK GL55 6LD (fax: +44-1386-842100; e-mail: psr@campden.co.uk).

Modelling flow patterns

The performance of baking ovens[7,8] was simulated using a CFD model of air flow within an oven to quantify heat transfer to, and mass transfer from, the product surface. The model was used to optimize the design of an experimental oven. Using such a CFD model, it was clear that the design of an oven could be optimized to ensure the correct distribution of heat in the oven to maximize the product quality. The inclusion of mass transfer (e.g. moisture loss) in the model allowed changes in bread quality during baking to be estimated.

CFD has been used by several researchers for the prediction of air distribution and temperature patterns inside chillers and retail display cabinets[9]. One particular example is for air flow prediction in meat chillers[10], where a two-dimensional model was used to aid in the design. A reasonable comparison, within 25%, was found between predictions and measured data. An extension to such work, which is feasible using CFD, is to include heat transfer in the problem. It is possible using current CFD techniques to include heat transfer between the product and the air and also to include radiation effects (i.e. loss of heat through imperfectly insulated chiller walls). This will assist in the design of chillers to ensure that the correct cooling regime is selected to give the desired rate of cooling for a particular product. It is also possible to include the effects of phase changes on heat transfer rates in a CFD model, thus enabling freezing operations to be modelled.

The use of CFD for the prediction of air flow movement, although not trivial, can be considered as the 'bread and butter' of CFD. To exploit CFD fully in food processing, it must also be capable of predicting flow patterns for liquids inside food processing equipment. CFD models have been developed for the flow of Newtonian fluids (e.g. glucose in pipelines)[6] and of non-Newtonian foods (e.g. ketchup) in a variety of continuous-flow processing equipment[5]. The models were developed to demonstrate CFD techniques to the food

PII: S0924-2244(97)01028-5

Box 1. Summary of the main types of fluid flow problems that general-purpose CFD codes can solve

Types of flow:

- Steady-state or transient
- Viscous or inviscid
- Laminar or turbulent
- Compressible or incompressible
- Subsonic, ultrasonic or supersonic
- Two-phase (continuous or particles)
- Chemical reaction
- Combustion
- Swirling
- Non-Newtonian (inelastic)

Modes of heat transfer:

- Convection
- Conduction
- Radiation

Types of material:

- Fluid (liquid or gas)
- Solid (homogeneous or porous)

Co-ordinate systems:

- Cartesian
- Cylindrical
- Body-fitted
- Moving and/or rotating

industry and have been, as far as possible, experimentally validated using a variety of process measurements including flow rate, pressure drop and velocity profile. The work has shown the importance of using the correct physical properties for the prediction of flow patterns for foods. Several non-Newtonian flow models, including the power-law model and the Casson model, were required to represent adequately the flow behaviour of a variety of types of foods.

Monitoring phase change

Many food processes require that the food undergoes a change of phase. A common example of this is the drying of foods. A particular example is the manufacture of powdered products such as instant coffee granules using spray driers. To model such a drier using CFD, it is necessary to make predictions of turbulent air flow movement, coupled with heat and mass transfer between the air and the coffee solution that is being dried. Such models can give predictions of particle-size and residence-time distributions and particle trajectories in driers to aid in their design and scale-up[11].

Other items of processing equipment that have been modelled using CFD include mixers[12] and pumps (I.S. Hamill *et al.*, unpublished), particularly centrifugal pumps (M.K. Yates *et al.*, unpublished). Although these examples are not from the food industry, the lessons learned from such predictions can be used in the food industry and the techniques applied to the mixing and pumping of foods.

A common application of CFD, and of numerical simulation in general, is in the prediction of flow patterns and temperatures during the thermal processing of foods. Particular examples of the application of CFD for this purpose include prediction of the transient temperature and velocity profiles in canned foods sterilized in a batch retort[13] and during the pasteurization of bottled beer[14]. The aims of such CFD simulations were to predict temperature patterns in the containers and thus locate the point of slowest heating to ensure that the product received a sufficient thermal process to render it commercially sterile.

It is clear from the reported uses of CFD in the food industry that several different applications have been studied using the technique. The general aim of all of the studies was to optimize existing processes and aid the process engineer in the design of new processes. It can therefore be concluded that CFD should be considered as an engineering tool whose application can assist in the efficient operation of a wide range of food processes.

To emphasize the importance that is being placed on CFD by the food industry and in particular by the UK government, a two-year Carrier Technology project, funded by the Department of Trade and Industry, was undertaken by three of the UK's leading food research organizations to demonstrate the application of CFD to three different process operations. The Campden & Chorleywood Food Research Association investigated the flow of non-Newtonian foods in continuous-flow systems[5,6]; the Food Refrigeration & Process Engineering Research Centre (the University of Bristol), the performance of retail refrigerated food display cabinets[9]; and the Leatherhead Food Research Association, the design of driers and ovens[4]. Each of the demonstration projects was guided by an industrial consortium to ensure the industrial relevance of the results.

The project demonstrated that a wide range of food industry applications can be studied and improved using CFD, although there are costs associated with such benefits. However, the biggest cost of undertaking any CFD activity was not that of the code and hardware but the cost of the CFD user, in that the learning curve is 1–2 years, from novice to proficient user. It was apparent that to utilize CFD to its fullest required personnel to be dedicated to CFD activities.

Having discussed the applications of CFD to be found in the food industry, it is worth considering what CFD is and what is available to any potential CFD user.

Commercial CFD codes

Many CFD codes are available commercially and many more reside in academic institutes. Of the commercial codes, some are general-purpose codes, such as CFX-4 (Computational Fluid Dynamics Services, Harwell, UK), PHOENICS (Cham Ltd, London, UK), and FIDAP and FLUENT (Fluent Europe Ltd, Sheffield, UK), offering solutions to a wide range of fluid flow and heat transfer problems. Box 1 summarizes the main features of such codes. The rest of the codes are application specific, such as POLYFLOW (Fluent Europe Ltd, Sheffield, UK), which was developed for the prediction of polymer melt flows, a problem requiring the solution of non-Newtonian viscoelastic flow behaviour.

Hardware requirements and software costs

All CFD codes are computer intensive, originally requiring workstation or mainframe computer facilities to run them; however, recent developments in PC technology have resulted in some of the codes becoming available for use on PC-based systems. The cost of a workstation and CFD code may be in the region of £40 000–50 000, whereas a PC-based system would probably cost less than £10 000. There is, however, a large difference in the performance of the two types of systems; workstation-based CFD codes are able to deal with larger and more complex problems and to solve them more quickly than PC-based CFD codes.

The use of any CFD code can be considered to consist of three main stages. The first is the definition of the problem; the second is its solution; and the third is the analysis of the results. This 'guided tour' of a generic CFD code considers the setting up, solution and analysis of a CFD problem.

In the definition of a CFD problem, three discrete elements need to be considered. The first is the transport equations that describe fluid flow. The second is a geometrical representation of the system being studied. The final component is the numerical procedure used to solve the transport equations in the geometry of interest.

The transport equations and related physics

The flow of any fluid can be described using transport equations that describe the conservation of mass (continuity), momentum and energy as the fluid flows (Box 2). They are derived by considering mass, momentum and energy balances in an element of fluid as it flows; from these the appropriate partial differential equations are derived. They are completed by adding two algebraic equations from thermodynamics: the equation of state and the constitutive equation.

Equation of state

The equation of state relates the density of a fluid to its thermodynamic state (temperature and pressure). Examples of such equations[15] include the ideal gas law for ideal gases and the van der Waals and Redlich–Kwong equations for non-ideal gases. A commonly used assumption for buoyancy problems is the Boussinesq buoyancy approximation[2], whereby all fluid properties are assumed to be constant except for the density.

Box 2. The transport equations and related physics of CFD

Conservation of mass equation:

$$\begin{pmatrix}\text{rate of change}\\ \text{of mass per}\\ \text{unit volume}\end{pmatrix} = \begin{pmatrix}\text{rate of mass flow}\\ \text{into unit volume}\end{pmatrix}$$

$$\frac{\partial \rho}{\partial t} = -\nabla.(\rho \mathbf{U})$$

Momentum equation:

$$\begin{pmatrix}\text{rate of}\\ \text{change of}\\ \text{momentum}\end{pmatrix} = \begin{pmatrix}\text{rate of}\\ \text{accumulation}\\ \text{of momentum}\\ \text{by convection}\end{pmatrix} + \begin{pmatrix}\text{rate of}\\ \text{accumulation}\\ \text{of momentum}\\ \text{by molecular}\\ \text{transfer}\end{pmatrix} + \begin{pmatrix}\text{sum of}\\ \text{forces}\\ \text{acting on}\\ \text{the system}\end{pmatrix}$$

$$\frac{\partial \rho \mathbf{U}}{\partial t} = -\nabla\cdot(\rho \mathbf{U} \otimes \mathbf{U}) + \nabla\cdot\sigma + \mathbf{B}$$

where σ is the stress tensor given by $\sigma = -p\delta + \mu[\nabla \mathbf{U} + (\nabla \mathbf{U})^T]$, and $\mathbf{B} = \rho \mathbf{g}$ for the Boussinesq approximation.

Energy equation:

$$\begin{pmatrix}\text{rate of}\\ \text{change}\\ \text{of energy}\end{pmatrix} = \begin{pmatrix}\text{rate of accumulation}\\ \text{of internal and kinetic}\\ \text{energy by convection}\end{pmatrix} + \begin{pmatrix}\text{rate of}\\ \text{accumulation}\\ \text{by}\\ \text{conduction}\end{pmatrix} + \begin{pmatrix}\text{rate of work}\\ \text{done by}\\ \text{surroundings}\\ \text{on system}\end{pmatrix}$$

$$\frac{\partial \rho H}{\partial t} = -\nabla\cdot(\rho \mathbf{U} H) + \nabla\cdot(\lambda \nabla T) + \frac{\partial p}{\partial t}$$

where H is the total enthalpy, given in terms of static (thermodynamic) enthalpy, h, by $H = h + \frac{1}{2}\mathbf{U}^2$

Equation of state (Boussinesq approximation):

$$\rho = \rho_0[1 - \beta(T - T_0)]$$

Nomenclature

$\mathbf{B}$ = body force (Pa)
$\mathbf{g}$ = acceleration due to gravity ($m{\cdot}s^{-2}$) – NB this is a vector quantity (0,9.81,0)
h = static enthalpy (J)
H = total enthalpy (J)
p = pressure (Pa)
t = time (s)
T = absolute temperature (K)
T_0 = reference temperature in Boussinesq buoyancy approximation (K)
$\mathbf{U}$ = velocity vector ($m{\cdot}s^{-1}$)
β = volumetric expansion coefficient
δ = Kronecker delta: $\delta_{mn} = 1$ for $m = n$ and $\delta_{mn} = 0$ for $m \neq n$
λ = thermal conductivity ($W{\cdot}m^{-1}{\cdot}K^{-1}$)
μ = molecular viscosity (Pa·s)
ρ = density ($kg{\cdot}m^{-3}$)
ρ_0 = reference density in Boussinesq buoyancy approximation ($kg{\cdot}m^{-3}$)
σ = stress tensor (Pa)

Vector and tensor operations

∇ = del, the vector operator. In rectangular co-ordinates, this is given by:

$$\nabla = \mathbf{i}\frac{\partial}{\partial x} + \mathbf{j}\frac{\partial}{\partial y} + \mathbf{k}\frac{\partial}{\partial z}$$

$\nabla.\mathbf{V}$ = divergence of a vector, defined as: $\nabla.\mathbf{V} = \frac{\partial V_i}{\partial x_i}$
$\otimes$ = tensor product, defined as: $(\mathbf{A}\otimes\mathbf{B})_{ij} = A_i B_j$
$\mathbf{A}^T$ = transpose of a tensor $\mathbf{A}$; if $\mathbf{A} = (A_{ij})$, then $\mathbf{A}^T = (A_{ji})$

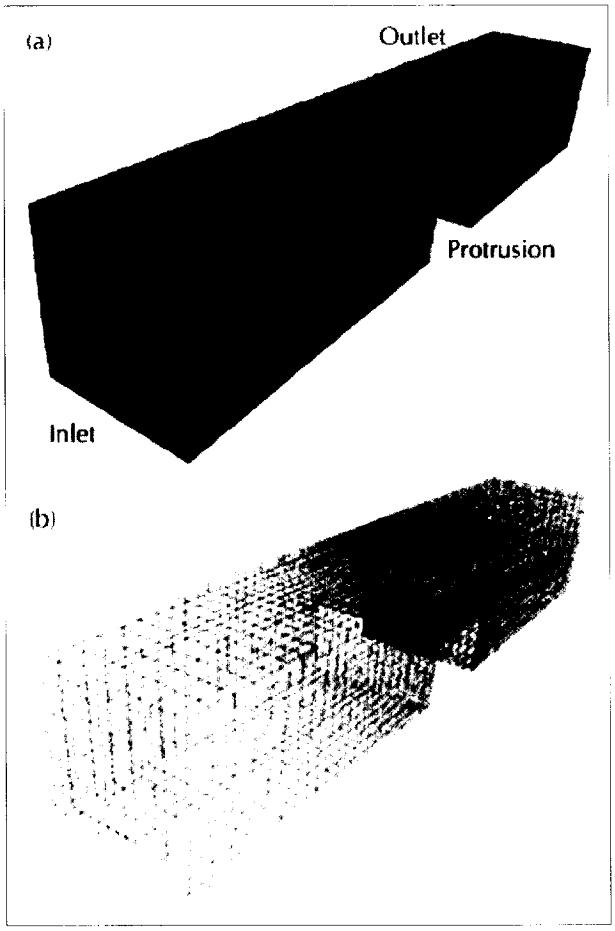

Fig. 1
Defining the problem: the physical space of the flow geometry being studied (a) is divided into a large number of small volumes, usually cubes, forming the 'computational mesh' (b). Although CFD calculations that use finer meshes generally yield more accurate predictions, they take longer to compute. (G.M. Scott, unpublished.)

Constitutive equation

The constitutive equation relates the static enthalpy of a fluid to its thermodynamic state. A discussion of such relationships can be found in any standard thermodynamics textbooks such as Smith and Van Ness[15].

Body forces

The body force, **B**, depends on the type of flow. For the Boussinesq buoyancy approximation, the body force becomes $\mathbf{B} = \rho \mathbf{g}$ (see Box 2 for nomenclature). Other body forces could include rotation forces (Coriolis and centrifugal), electrostatic forces and resistances, such as that imposed on a fluid as it flows through a porous medium.

These balances and equations are presented in Box 2 and discussed in more detail in Bird *et al.*[16]

Turbulence

The transport equations presented can be applied to both laminar and turbulent flow conditions. Where turbulent flow conditions prevail, a suitable turbulence model is required to describe the turbulence and its influence on flow conditions. Several turbulence models are available in most CFD codes that attempt to solve the time-dependent nature of turbulent flow[17]. It is beyond the scope of this review to discuss turbulence models in detail except to mention some of the commonly used ones. The models that are used fall into two broad classes: eddy viscosity models and second-order closure models[18]. Eddy viscosity models include the k-ε model and variations on it. Second-order closure models include the differential stress model and the differential flux model.

The inclusion of turbulence in a CFD problem makes its solution more complex because extra equations need to be solved.

Non-Newtonian flow behaviour

When considering the flow of food products, it is often necessary to take the rheological nature of a food into account because this will dictate its flow behaviour. Most foods exhibit some form of non-Newtonian behaviour and many different flow models have been used to describe such behaviour[19]. In general, any type of time-independent inelastic flow model can be included in most general-purpose CFD codes and, in some cases, these models are standard options. In those cases where more complex flow behaviour exists, such as time-dependent flows, more specialized CFD codes need to be used; these tend to be problem specific and are not generally available in the public domain.

As for turbulence problems, the inclusion of a non-Newtonian flow model requires the solution of an extra equation, that for viscosity. It should be noted that the current state of CFD does not permit the simultaneous prediction of turbulent flow for non-Newtonian fluids because this is at the forefront of numerical non-Newtonian rheology research.

Flow geometry and boundary conditions: defining the problem

The equations describing the motion of a fluid are solved for the geometry, or flow domain, being studied. Most modern CFD codes are capable of modelling quite complex geometries[2], such as the internals of a combustion engine or pump. The physical space of the geometry is split into a large number of small volumes, which are normally topologically cubic. Each of the volumes is connected to its neighbours, forming the computational mesh, such that the conservation of mass, momentum and energy is obeyed. In general, the smaller the volumes, or the finer the mesh, the more accurate the CFD predictions will be; however, this increase in accuracy is accompanied by greater demands on computing resources and longer solution times.

The generation of the computational mesh is normally performed by graphical pre-processors, which are either part of the CFD code or third-party software, whereby the geometry of interest is defined using a suitable coordinate system (e.g. Cartesian or cylindrical). Figure 1a shows an example of the simple flow geometry for a square duct containing a protrusion. Once the geometry has been defined, it is then split into a number of volumes that form the computational mesh. Figure 1b shows the computational mesh for the square duct problem. Complex flow geometries are meshed using a body-fitted co-ordinate system, which allows the cell volumes to distort and thus take on the shape of the flow geometry.

In any CFD problem, the definition of boundary conditions is essential because these will define the flow conditions inside the flow domain. In general, any inlets or outlets to the flow domain need to be defined and appropriate boundary conditions set. For example, in the case of an inlet to the domain, the velocity or flow rate of a fluid and its temperature (for non-isothermal models) may be set. Outlets can be defined using, for example, flow rate, velocity or pressure. Other types of boundary conditions that may need to be set are those for walls and solids, particularly if they act as heat transfer boundaries.

After defining the computational mesh and boundary conditions, the user needs to define the assumptions made in the model, for example whether the flow is laminar or turbulent and whether heat transfer takes place. This usually involves writing a command file that contains these assumptions along with physical properties and any other information required by the CFD solver, such as the number of iterations and which turbulence model to use.

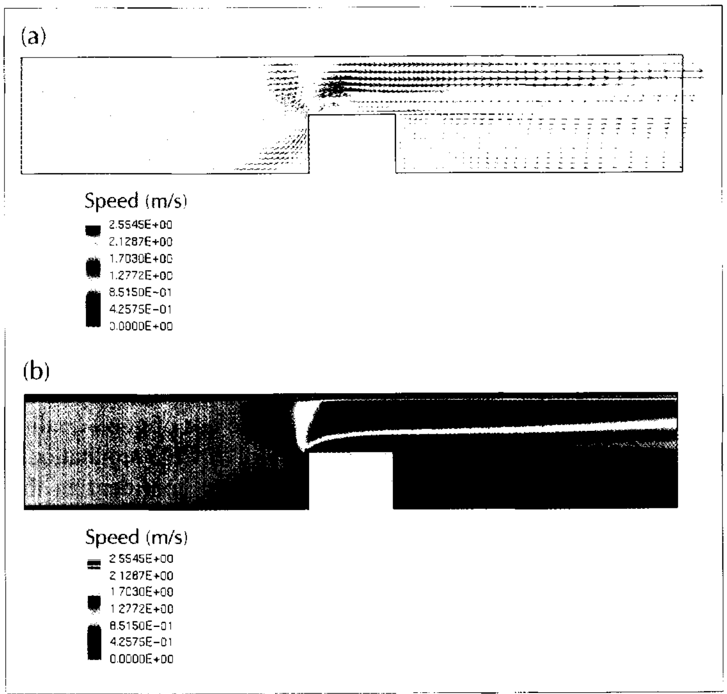

Fig. 2
Presenting the results: these are usually examined graphically in the form of a vector plot (a), a contour plot (b), or an iso-surface plot (not shown). (G.M. Scott, unpublished.)

Numerical procedures: solving the problem

The solution of the transport equations for the geometry under study is not a trivial matter and cannot be solved readily, if at all, by using analytical techniques. CFD uses numerical techniques to solve discretized representations of the transport equations[2]. Different CFD codes use different numerical techniques. Direct methods, which can be both extremely accurate and rapid, can be used if sufficient computing power is available; however, they sometimes have difficulty finding a solution. Many codes, therefore, use iterative methods to solve the equations because they tend to be more robust, although they can take longer to converge.

Standard texts are available that provide good background material on numerical simulation[7] and CFD[8].

CFD results: files and pictures

The results from a CFD run can be examined in two ways. The first method, which is probably the traditional method, is to read through the output file from the code. The output file can contain all sorts of information including the spatial co-ordinates of all of the cells in the computational mesh and the solved transport variables of velocity, pressure, temperature, etc., for each cell. In the case of a large CFD problem, say, greater than 100 000 cells, it is obvious that the user does not want to read through this file. Thus, the second method offers the user the ability to visualize the results. This method, often referred to as post-processing, takes the results from the CFD solver and allows the user to display variables graphically on the computer screen, for all or part of the flow domain. Graphical presentation methods include vector plots (scaled arrows pointing in the direction of flow), contour plots on a two-dimensional slice through the domain and iso-surface plots (a three-dimensional surface on which a variable is constant). Figure 2a shows velocity vectors on a two-dimensional slice through the centre of a square duct, whereas Fig. 2b shows a two-dimensional contour plot of speed through the centre of the duct.

Conclusions: the future of CFD in the food industry

It is clear that the food industry is starting to take steps to accept CFD as an engineering tool. It has been shown that the results from a CFD study can assist in understanding the dynamics and the underlying physics of a processing operation and thus aid in the optimization and design of existing and new processing equipment. Currently, the benefits of CFD are often overshadowed by the investment required to adopt this technique. CFD is an expensive tool, requiring considerable investment in personnel time, CFD code and hardware, such that, in the current economic climate, most food companies cannot afford to invest in CFD technology. To assist in this dilemma, food companies could consider sub-contracting work out to a suitable research organization, one that is both a CFD user and an expert in food processing. However, the real costs of CFD should decrease with time, to the point where CFD will become financially viable for many companies, assuming the company can support the personnel costs incurred and lead time required before seeing a return on its investment.

However, many current technical obstacles have to be overcome to ensure the acceptance of CFD by the food industry. The biggest is, perhaps, the issue of confidence in the predictions. As with any modelling technique, the results are only as good as the data fed into the model. It is known that high-quality thermo-physical property data for food materials can be difficult to generate. It would, therefore, be beneficial, in the short term, for the industry to pool all of the available data, say, in a physical property database, so that they are accessible for the good of all. The development of a food properties database is currently high on the agenda of both the UK Ministry of Agriculture, Fisheries and Food, and the European Union. The use of good-quality data can only increase the standard of CFD simulations, ensuring that industry becomes more confident in the technique.

It is envisaged that codes will become easier to use, reducing the lead time to benefits. CFD codes will have post-processors that are self-meshing, whereby the user defines the geometry of interest and the code generates the computational mesh. This feature should be available on the next release of the CFX code. As the number of CFD users in the food industry increases, so industry 'customized' versions of the codes should appear. These could contain a library of standard processing items, in which geometries are already defined, requiring the user to define the appropriate boundary conditions. Input to and output from the codes will be described using food processing terminology, ensuring that the whole industry can benefit from CFD and not just the process engineer who is sitting in front of the computer. CFD predictions could be enhanced for the food scientist by the inclusion of microbiological models of death kinetics, which would utilize CFD's transient predictions of temperature profiles. This would enable processes where microbiological activity is important to be optimized in terms of, for example, integrated lethality, F_0, or cook value, C, as well as flow and temperature profiles.

We believe that CFD is a powerful engineering tool that will greatly benefit the food industry in its continual quest for process and product improvement. The demonstration of CFD to the food industry has happened; it is now up to the industry whether it adopts it or not.

References

1 Collins, M.W. and Ciofalo, M. (1991) 'Computational Fluid Dynamics and Its Application to Transport Processes' in *J. Chem. Technol. Biotechnol.* 52(1), 5–47

2 Quarini, G.L. (1995) 'Applications of Computational Fluid Dynamics in Food and Beverage Production' in *Food Sci. Technol. Today* 4, 234–236

3 Scott, G.M. (1994) 'Computational Fluid Dynamics for the Food Industry' in *Food Technol. Int. Eur.* 1994, 49–51

4 Janes, D.A. and Dalgety, E.F. (1996) *How to Use CFD in the Food Industry*, Leatherhead Food Research Association Technical Report, Leatherhead, UK

5 Scott, G.M. (1977) *Simulation of the Flow of Non-Newtonian Foods Using Computational Fluid Dynamics*, Campden & Chorleywood Food Research Association R&D Report No. 34, Chipping Campden, UK

6 Scott, G.M. (1996) *Computational Fluid Dynamics – Modelling the Flow of Newtonian Fluids in Pipelines*, Campden & Chorleywood Food Research Association R&D Report No. 24, Chipping Campden, UK

7 Velthuis, H., Dalhuijsen, A. and de Vries, U. (1993) 'Baking Ovens and Product Quality' in *Food Technol. Int. Eur.* 1993, 61–66

8 de Vries, U., Velthuis, H. and Koster, K. (1995) 'Baking Ovens and Product Quality – A Computer Model' in *Food Sci. Technol. Today* 4, 232–234

9 Foster, A.M. (1996) *Carrier Technology Programme: Demonstration of CFD to the Food Industry – 1st Year Report*, Food Refrigeration and Process Engineering Research Centre Report, Langford, Bristol, UK

10 Mirade, P.S., Daudin, J.D. and Arnaud, G. (1995) 'Two-dimensional Simulation of the Air Flow in Two Industrial Meat Chillers' in *Int. J. Refrig.* 18(6), 403–412

11 Oakley, D.E. (1994) 'Scale-up of Spray Dryers with the Aid of Computational Fluid Dynamics' in *Drying Technol.* 12(1/2), 217–233

12 Weetman, R.J. (1996) 'Simulation Helps Quadruple Mixer Life in Abrasive Conditions' in *Chem. Technol. Eur.* 3(1), 17–20

13 Kumar, A. and Bhattacharya, M. (1991) 'Transient Temperature and Velocity Profiles in a Canned Non-Newtonian Liquid Food During Sterilization in a Still-cook Retort' in *Int. J. Heat Mass Transfer* 34(4/5), 1083–1096

14 Engleman, M.S. and Sani, R.L. (1983) 'Finite Element Simulation of an In-package Pasteurization Process' in *Numer. Heat Transfer* 6, 41–54

15 Smith, J.M. and Van Ness, H.C. (1975) *Introduction to Chemical Engineering Thermodynamics*, McGraw-Hill International

16 Bird, R.B., Stewart, W.S. and Lightfoot, E.N. (1960) *Transport Phenomena*, John Wiley & Sons

17 Coulson, J.M. and Richardson, J.F. (1977) *Chemical Engineering Volume 1*, Pergamon Press

18 Computational Fluid Dynamics Services (1995) *CFX 4.1 Flow Solver User Guide*, Computational Fluid Dynamics Services, Harwell, UK

19 Holdsworth, S.D. (1993) 'Rheological Models Used in the Prediction of the Flow Properties of Food Products: A Literature Review' in *Trans. Inst. Chem. Eng.* 71(C4), 139–179

Cross-disciplinary trends

The following recently published articles from our 'sister' journals may be of interest to TIFS readers:

Antioxidant properties of phenolic compounds, by Catherine A. Rice-Evans, Nicholas J. Miller and George Paganga, *Trends in Plant Science* 2(4), 152–159

Recent advances in polymers for barrier applications, by H. Craig Silvis, *Trends in Polymer Science* 5(3), 75–79

Polymer membranes for use in food processing, by Xingwu Wang and H. Garth Spencer, *Trends in Polymer Science* 5(2), 38–39

Campylobacter–host cell interactions, by Karl G. Wooldridge and Julian M. Ketley, *Trends in Microbiology* 5(3), 96–102

Interactions between enteropathogenic *Escherichia coli* and host epithelial cells, by Michael S. Donnenberg, James B. Kaper and B. Brett Finlay, *Trends in Microbiology* 5(3), 109–114

Biotechnology and the red seaweed polysaccharide industry: status, needs and prospects, by Donald Renn, *Trends in Biotechnology* 5(1), 9–14

Molecular biology and pathogenesis of prion diseases, by Stanley B. Prusiner, *Trends in Biochemical Sciences* 21(12), 482–487