[Volume 4 issue 07 July 2016]
PageNo.1543-1553 ISSN : 2320-7167

INTERNATIONAL JOURNAL OF MATHEMATICS AND COMPUTER RESEARCH

A Review on Computational Fluid Dynamics (CFD) Modelling of Refrigerated Space

Arun Prasath Venugopal , S. Ganapathy and Aarthy Viswanath
Department of Food & Agricultural Process Engineering,
Tamil Nadu Agricultural University.
Email: arun16foodengg@gmail.com

Abstract

Computational Fluid Dynamics (CFD) modelling is the process of representing a fluid flow problem by mathematical equations based on the fundamental laws of physics, and then solving those equations numerically using computer to predict the variation of the relevant parameters within the flow field. The recent adoption of CFD has been both inevitable and progressive, as the high costs and time consumption associated with experimentation has often precluded the desire to produce efficient in-depth results. Moreover, the assumptions, generalizations and approximations associated with analytical models have swayed their reduction in the development of flow solutions. By considering these limitations coupled with recent achievements in the development of numerical solutions for the Navier-Stokes equations and the amelioration of computing power and efficiency, it is easy to understand why confidence has both increased and advanced the application of CFD as a viable alternative in the field of refrigeration.

Key words: Computational Fluid Dynamics (CFD), Modelling, Simulation, Refrigerated space.

Introduction

Computational fluid dynamics (CFD) was originally developed from the pioneering accomplishments of enthusiasts, who in their endeavours to procure insight into fluid motion instigated the development of powerful numerical techniques that have advanced the numerical description of all types of fluid flow (Hoang *et al.*, 2000). CFD is now maturing into a powerful and pervasive tool in many industries, with each solution representing a rich tapestry of mathematical physics, numerical methods, user interfaces, and state-of-the art visualisation techniques (Qian, 2002). So great has the impetus been to propel CFD that it is now used as much as the traditional didactic methods of experimentation and analytical modelling to solve fluid flow problems.

The links between CFD and the processes associated with the food and beverage industry such as mixing, drying, cooking, sterilization, chilling and cold storage are profound. Such processes are used regularly to enhance quality, safety and shelf life of foodstuffs (Qian, 2002). With direct benefits for both consumer and the natural environment, applications of CFD have become more widespread in the food industry. CFD research has meant that products can be processed and stored in more efficient systems. Furthermore, CFD can aid food companies to respond to an expanding marketplace by enhancing and developing processing strategies, whilst endeavouring to maintain high levels of product quality.

CFD applications in food processing

CFD applications in food industry may assist in a better understanding of the complex physical mechanisms. The general application of CFD to the food processing industry. Moreover, other literatures are also available on specific CFD application areas such as: Clean-room design, Refrigerated transport, Static mixers, and Pipe flow. Since CFD technique can be of great benefit to the food processing industry, fast development has taken place in the past few years. CFD, as a tool of research for enhancing the design process and understanding of the basic physical nature of fluid dynamics can provide benefits to the food processing industry in many areas, such as drying, sterilization, mixing, refrigeration and other application areas.

Types of Models

According to whether the modelled systems change with time or not, models can be classified as steady state and dynamic (unsteady-state).

A. Steady-state models

Steady-state models are suitable for modeling systems whose major parameters do not change with time. Steady-state models are also used to assess the performance of a system under different sets of operating conditions (Touber, 1984). Steady state modeling may be applied to describe time-averaged behaviour of a transient system, while heat and mass accumulation in the system is negligible. Steady-state models generally demand less computational time and a small amount of input data, as the time variability in system parameters is not considered.

B. Dynamic models

Dynamic models are applied to assess how the time-variable effects, such as heat load, environmental conditions and start-up transients, influence normal system operation; accordingly advanced control strategies or detailed controllers may be developed (Cleland & Cleland, 1989).

Dynamic models are usually considered closer to real world situations because most of them are time-dependent. However, as one more dimension (time) has to be dealt with mathematically, the dynamic modeling approach requires more input data (initial conditions) and more computational capacity while solving the models. Based on the approaches for modeling positional variation of variables (space discretisation), models are divided into zoned and fully distributed models.

C. Zoned models

In zoned models, the space to be modelled is divided into several zones. For each zone an ordinary differential equation is adequate for each variable as the conditions within the zone are assumed to be uniform. Movement of fluid within a region may be defined by a plug-flow pathway, and the position of zones is arranged along the flow pathway through the system in a sequential fashion (Amos, 1995).

Generally, only temperature and fluid concentration are solved in zoned models, while fluid velocity is defined with experimental data instead of by solving the momentum conservation equations. A single zoned model is the simplest case in which the whole calculation region is treated as uniform.

D. Fully distributed models

Fully distributed models are also called fluid dynamic models, and use partial differential equations (PDEs) to formulate the full position-variability. These PDEs describe heat, mass, and momentum conservation within the considered region. This approach is usually able to simulate the real world more accurately than the zoned models, since fewer assumptions are needed to develop a fully distributed model.

Finite-difference, finite-element, and finite-volume methods are the most commonly used numerical tools for solving the transport equations. Fully distributed modeling requires much larger computer memory and computational time than zoned modeling.

CFD Modelling of airflow patterns in refrigerated spaces

The application of CFD based models for studying airflow patterns in refrigerated spaces is still at the development stage. Unlike an office or living room, a refrigerated space is almost fully occupied with produce and packaging, and the free space for the air circulation has a very complicated geometry. Therefore, either a very fine or complex grid or the porous media approach has often been used to deal with the irregular geometry of the refrigerated space while solving the transport equations.

The modelling was carried out in two steps. The first step was to model the flow pattern without considering any heat and mass transfer. Because the airflow was treated as steady-state (the effect of buoyancy was also neglected), the three-dimensional Navier Stokes equation was decoupled from the equations for energy and mass transfer. The air velocity and turbulence quantities in the k-s model were solved using the commercial CFD package PHOENICS.

The second step was to model the heat and mass transport based on the predicted flow pattern. Although such a strategy reduced the computational time significantly, the prediction of the airflow pattern in the room still required 100 h of computing on workstation. The produce in the refrigerated room was modelled as a porous medium. The model was tested against measured temperatures, and satisfactory results were obtained.

Van Gerwen *et al.,* (1991) used the PHOENICS package to simulate stationary 3D airflow distribution in a carcass chiller. The carcass rows in the chiller were modelled as porous layers. The air velocity around a carcass, calculated by the CFD model, was used as input for calculating heat and mass transfer coefficients on the surface of a thermal carcass model. Their model was also validated by measured data, and good agreement was found. Mariotti *et al.*, (1995) used a similar approach to model air distribution in a refrigerated room. The velocity field was firstly solved under a steady state assumption. Then using the calculated velocity distribution, the transient temperature field was solved. The solution procedure was based on the finite element method, which was claimed to provide the intrinsic flexibility to treat complex flow situations and irregular geometry conditions.

CFD Modeling of heat transfer during product cooling

Product cooling involves several heat generation and transfer processes including:

- Heat generation due to respiration
- Convection on the surfaces of products and packaging materials
- Conduction within products, between products, and between products and packaging materials
- Convection and conduction within the cooling medium
- Radiation between surfaces of products and packaging materials
- Evaporative cooling effect due to transpiration

The influences of the above processes on cooling efficiency differ with cooling conditions. For instance, the effects of transpiration and respiration may be negligible during product precooling due to high cooling rates. However, these two factors have been shown to play an important role in product thermal stability during long term storage.

Based on the approaches in handling the cooling media, the relevant heat transfer models are divided into two categories. In the first category, only heat conduction within the product is modelled in detail and the temperature of cooling medium is assumed to be constant or a function of time and no differential equation is derived for the energy conservation of cooling medium.

The models in this category are usually used to simulate single product situations. The second category models take into account the energy conservation for both product and cooling medium, and generally two differential equations are needed to simulate the temperature variation within the product and cooling medium. Produce contained in packages is often modelled with this approach.

Product heat conduction plus cooling media models

The heat conduction models are most suitable for modelling heat transfer for single product items. During cooling, most products are packed or bulk-stacked, and the temperature and velocity distribution within the cooling media may be significantly affected by packaging and stack patterns. In these cases, a simple conduction model may not be practical.

The air temperature and product (heat and mass transfer) characteristics were assumed not to vary within each layer. The airflow within the bulk store was described as Darcy flow through a porous medium. The Boussinesq approximation was applied to describe natural convection. A similar approach was adopted by (Bazan *et al.*, 1989) to predict the three-dimensional temperature response during room cooling of a confined bin of spherical fruit. Close agreement between simulation and experimental results was obtained.

Amos (1995) developed a multi-zone model for predicting apple temperature and weight loss with both position and time within a ventilated carton. In the model, airflow inside the carton was modelled by defining forced convection pathways with natural convection mixing to adjacent zones. The air in each zone was assumed perfectly mixed. Energy and water vapour mass balances were performed on each zone to determine air enthalpy and humidity ratio, as well as the temperature of apples and packaging materials. The model predictions fitted the measured temperature data satisfactorily. However, the airflow pattern was estimated from measured air velocity data within the specified carton.

Qian (1998) developed a CFD model to simulate airflow patterns and heat transfer in a ventilated apple carton during precooling. The CFD package PHOENICS was used to solve the model. The flow equations were solved under steady-state condition for both laminar and turbulent situations. Based on the predicted airflow patterns, the energy equations were solved dynamically to obtain temperature profiles. The temperatures in the centres of apples in various positions were measured.

Procedure for CFD Modelling of refrigerated space

The mathematical models is defined as the equations that simulate real world situations, because they behave in a manner analogous to the actual situation. Application of mathematical models reduces the cost of experimentation, as modelling allows more alternatives to be considered which may be difficult or expensive to test.

The identified five main stages in a modelling process are asking the questions, selecting a modelling approach, formulating the model, solving the model and answering the questions. The starting steps are to examine the real world system to be modelled, and to identify the problems to be solved. These would enable to decide on the objectives of modelling, the required accuracy and the type and size of computer envisaged. The general system for equation development suitable for modelling in the area of refrigeration is summarized in Fig.1.

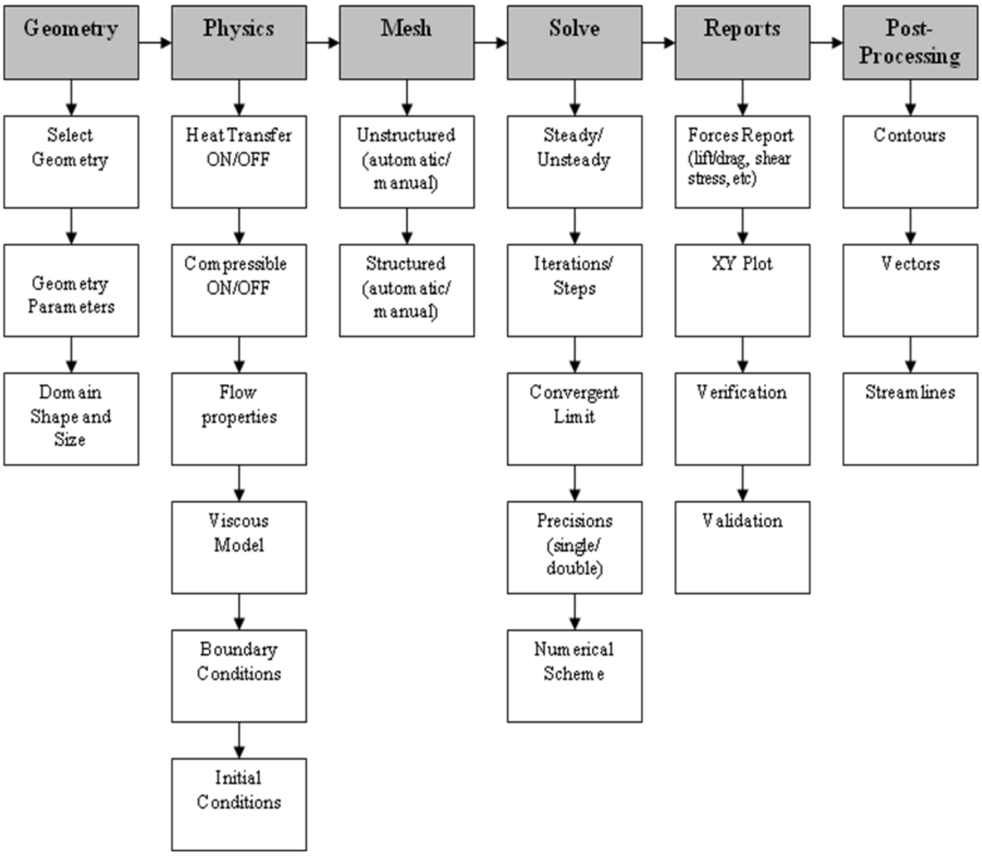

Fig.1. Flow Diagram of Computational Fluid Dynamics

Governing Equations

The starting point of CFD is the fundamental equations of fluid dynamics that describe the transport phenomena based on the conservation laws. All the conservation equations, also known as field equations, represent the variation of solution variables in space and time. These basic equations for incompressible, viscous flow and the energy equation for solid are presented as follows,

A. Continuity equation

The fluid density for incompressible flow was calculated from the empirical formula available in the standard data book for the flowing fluid temperature and pressure (Wang and Touber, 1990).

$$\nabla.(\rho_f U) = S_m \qquad \text{-------------- (1)}$$

B. Momentum equation

The governing equation based on the conservation of momentum of a Newtonian fluid flow and applied to an infinitesimal small volume in a Cartesian co-ordinate system (x; y; z) is, (Wang and Touber, 1990).

$$\nabla.(\rho_f U v_j = -\nabla p + \nabla.\left(\mu_{ef}\left[\nabla v_j\right]\right) + \rho_f \vec{g} + S \qquad \text{---------- (2)}$$

C. Energy equation

The energy equation was used to describe the heat transfer inside the precooling chamber. (Wang and Touber, 1990).

$$\nabla.\left(U\left(\rho_f E_f + P\right)\right) = \nabla.\left[k_{eff}\nabla_T - \left(\sum_i h_i \vec{J}_i\right)\right] + S_f^h \qquad \text{---------- (3)}$$

The energy equation for convection heat transfer between air and produce was modelled according to Newton's law of cooling.

$$q_{air-produce} = -h_t(T_s Surface - < T_a > a) \qquad \text{---------- (4)}$$

In the above mentioned governing equations, U (v_1, v2, v3) is the velocity vector, consisting of the three components v1, v2, v3 (m/s), p is the pressure (Pa); T is temperature (^{o}C); ρ density (kg/m^3); μ Viscosity (kg/ms)

Where, the μ_T and μ_{eff} are the turbulent and effective viscosity, which are needed for the turbulence model. Hence the turbulent flow is experienced throughout the domain except porous region, k and ε (realizable) turbulence model was used to model the turbulence.

Momentum equations for porous media

The porous media models for single phase flows use the superficial velocity Porous formulation as the default. In ANSYS FLUENT the superficial phase or mixture velocities calculations are based on the volumetric flow rate in a porous region. The porous media model is described in the following sections for single phase flow.

- In the Eulerian model, the general porous media modelling approach, physical laws, and equations described below are applied to the corresponding phase for mass continuity, momentum, energy, and all the other scalar equations.

- The Superficial Velocity Porous Formulation generally gives good representations of the bulk pressure loss through a porous region. However, since the superficial velocity values within a porous region remain the same as those outside the porous region, it cannot predict the velocity increase in porous zones and therefore limits the accuracy of the model.

Porous media are modelled by the addition of a momentum source term to the standard fluid flow equations. The source term is composed of two parts: a viscous loss term (Darcy, the first term on the right-hand side of (Eqn.5.) and an inertial loss term (the second term on the right-hand side of (Eqn.5). (Xu and Burfoot, 1999)

$$Si = -\left(\sum_{j=1}^{3} D_{ij}\mu v_j + \sum_{j=1}^{3} C_{ij}\frac{1}{2}\rho|v|v_j\right) \qquad \text{---------- (5)}$$

Where, S_i is the source term for the *i*th (x, y, or z) momentum equation, |v| is the magnitude of the velocity and D and C are prescribed matrices. This momentum sink contributes to the pressure gradient in the porous cell, creating a pressure drop that is proportional to the fluid velocity (or velocity squared) in the cell. To recover the case of simple homogeneous porous media (Xu and Burfoot, 1999).

$$S_i = -\left(\frac{\mu}{\alpha} v_i + C_2\frac{1}{2}\rho|v|v_i\right) \qquad \text{--------- (6)}$$

Where, α is the permeability and C_2 is the inertial resistance factor.

Since the flow through the porous medium is assumed to be laminar, the pressure drop is typically proportional to velocity and the constant C_2 can be considered to be zero. Ignoring convective acceleration and diffusion, the porous media model then reduces to Darcy's Law:

$$\nabla_p = -\frac{\mu}{\alpha}\vec{v} \qquad \text{--------- (7)}$$

The pressure drop that ANSYS FLUENT computes in each of the three (x,y,z) coordinate directions within the porous region is then

$$\Delta p_x = \sum_{j=1}^{3} \frac{\mu}{\alpha xj} v_j \Delta n_x \qquad \text{--------- (8)}$$

$$\Delta p_y = \sum_{j=1}^{3} \frac{\mu}{\alpha yj} v_j \Delta n_y \qquad \text{--------- (9)}$$

$$\Delta p_z = \sum_{j=1}^{3} \frac{\mu}{\alpha zj} v_j \Delta n_z \qquad \text{---------- (10)}$$

Where, $1/\alpha ij$ are the entries in the matrix D in Eqn.5, v_j are the velocity components in the *x*, y, and z directions, and Δn_x, Δn_y, and Δn_z are the thicknesses of the medium in the *x*, y, and z directions (Xu and Burfoot, 1999).

Equilibrium Thermal Model Equation for Porous Media

For present simulations the porous medium and fluid flow are assumed to be in thermal equilibrium, the conduction flux in the porous medium uses an effective conductivity and the transient term includes the thermal inertia of the solid region on the medium

$$\frac{\partial}{\partial t}\left(\gamma\rho_f E_f + (1-\gamma)\rho_s E_s\right) + \nabla.\left(\vec{v}\left(\rho_f E_f + p\right)\right) = \left(S_f^h + \nabla.\left[k_{eff}\nabla T - \left(\sum_i h_i J_i\right) + \left(\bar{\bar{\tau}}.\vec{v}\right)\right]\right)$$

--- (11)

Where, E_f - Total fluid energy; E_s - Total solid medium energy; ρ_f - Fluid density; ρ_s - Solid medium density; γ - Porosity of the medium; κ_{eff} - Effective thermal conductivity of the medium and S_f^h - fluid enthalpy source term

The effective thermal conductivity in the porous medium k_{eff}, is computed by as the volume average of the fluid conductivity and the solid conductivity:

$$k_{eff} = \gamma k_f + (1-\gamma)k_s \qquad \text{---------- (12)}$$

Where, k_f = fluid phase thermal conductivity (including the turbulent contribution, k_t); k_s = solid medium thermal conductivity.

Advantages of Using CFD

CFD has grown from a mathematical curiosity to become an essential tool in almost every branch of fluid dynamics. It allows for a deep analysis of the fluid mechanics and local effects in a lot of equipment. Most of the CFD results will give an improved performance, better reliability, more confident scale-up, improved product consistency, and higher plant productivity (Donna *et al.*, 2008). Some design engineers actually use CFD to analyse new systems before deciding which and how many validation tests need to be performed. The advantages of CFD can be categorised as (Donna *et al.*, 2008):

- It provides a detailed understanding of flow distribution, weight losses, mass and heat transfer, particulate separation, etc. Consequently, all these will give plant managers a much better and deeper understanding of what is happening in a particular process or system.
- It makes it possible to evaluate geometric changes with much less time and cost than would be involved in laboratory testing.
- It can answer many ‘what if’ questions in a short time.
- It is able to reduce scale-up problems because the models are based on fundamental physics and are scale independent.
- It is particularly useful in simulating conditions where it is not possible to take detailed measurements such as high temperature or dangerous environment in an oven.

- Since it is a pro-active analysis and design tool, it can highlight the root cause not just the effect when evaluating plant problems.

Conclusion

Many food processing operations such as chilling, drying, cooling, mixing, freezing, cooking, pasteurization and sterilization rely on fluid flow. The transfer of CFD approaches to the food industry has provided food engineers new insight and understanding to the likely performance of food equipment at the design stage and confidence in the quality or safety of food products (Chourasia *et al.*, 2006). Equipment such as cooling chamber, heat exchangers, refrigerated display cabinets and spray dryers has been improved through the application of CFD techniques in aiding the understanding of their operation and design process. CFD has become a powerful tool in the development, trouble shooting and optimization of refrigeration equipment.

Reference

1. Amos, N.D. 1995. Mathematical modelling of heat transfer and water vapour transport in apple cool store. PhD Thesis, Massey University, Palmerston North, New Zealand.
2. Bazan, T., Chau, K.V. and Baird, C.D. 1989. Heat transfer simulation of the bulk cooling of fruits. ASAE Paper No. 89559.
3. Chourasia, M.K, and T.K. Goswami 2006, Steady state CFD modeling of airflow, heat transfer and moisture loss in a commercial potato cold store, International Journal of Refrigeration 30 : 672-689,
4. Cleland, D.J. and Cleland, A.C. 1989. Appropriate level of model complexity in dynamic simulation of refrigeration systems. Refrigeration. Science. & Technology, 1989-1: 26 1 -268.
5. Donna A. Calhoun, Christiane Helzel, and Randall J. LeVeque. 2008, Logically Rectangular Grids and Finite Volume Methods for PDEs in Circular and Spherical Domains, Society for Industrial and Applied Mathematics, Vol. 50, No. 4, pp. 723–752.
6. Hoang, M.L, P. Verboven, J. De Baerdemaeker, and B.M. Nicolai. 2000, Analysis of the air flow in a cold store by means of Computational Fluid Dynamics, International Journal of Refrigeration 23: 127-140
7. Mariotti, M., Rech, G. and Romagnoni, P. 1995. Numerical study of air distribution in a refrigerated room. 19th International Congress of Refrigeration 1995, 2: 98- 105.
8. Qian Zou. 1998. CFD Modelling of Airflow and Heat Transfer in a Ventilated Carton. Master thesis, Massey University, Palmerston North.n
9. Qian Zou. 2002, A CFD modelling system for air flow and heat transfer in ventilated packing systems during forced- air cooling of fresh produce, PhD thesis, Massey University.

10. Touber.S. (1984). Principles and methods for mathematical modelling the steady-state and dynamics behaviour of refrigeration components and installations, Refrigeration Science & Technology, 1 984-2: 163-1 75.
11. Van Germen, R.J.M., Van der Sluis, S.M. & Van Oort, H. (1991). Computer modelling of carcass chilling models, Proceedings of18th International. Congress of Refrigeration, 4: 1 893- 1 897.
12. Wang, H. and Touber, S. 1990. Distributed dynamic modelling of a refrigerated room. International Journal of Refrigeration, 13: 2 1 4-222.
13. Xu, Y. and Burfoot, D. 1999. Simulating the bulk storage of foodstuffs. Journal of Food Engineering, 39: 23-29.