CFD Simulation of Transport Phenomena in Dairy Processing Applications

Chopde SS*, Ankit Kumar Deshmukh, Mehta Sumit, Somveer, Parameswari PL
Ph. D. Research Scholar, Dairy Engineering Division, ICAR-National Dairy Research Institute, Karnal
*Corresponding author: santosh.der@gmail.com

Computational fluid dynamics (CFD) is a powerful tool for simulating fluid flow and heat transfer in various industries. It combines fluid mechanics, mathematics, and computer science to solve governing equations using computer software. Its application has gained global attention since the emergence of digital computers. CFD simulations offer advantages over empirical experiments. It allow exploration of any location in the region of interest and interpretation of performance using multiple thermal and flow parameters. Simulations are cost-effective compared to traditional experiments, as they provide engineering data for design without the need for costly physical testing. CFD enables modeling of various process conditions and simulation of difficult-to-test flow and heat transfer processes. It provides control over the physical process, allows isolation of specific phenomena for research, and can be performed more quickly than laboratory tests. Simulations facilitate early modifications in the design process.

Understanding transport phenomena in dairy processing is crucial for process analysis, prediction and design. CFD is a valuable tool for solving complex problems related to momentum, heat, and mass transfer. Visualizing simulation results through attractive color figures and animations aids in interpreting physical phenomena, improving process and product quality.

2. Stages in performing a CFD analysis

To conduct a CFD analysis, the analyst formulates the problem mathematically, utilizes CFD software to represent the problem scientifically, and performs calculations using the computer. The resulting data is then examined and interpreted by the analyst. Three key steps are involved in CFD simulation.

Pre-processing: All the tasks that take place before the numerical solution process are called pre-processing. This includes problem thinking, meshing and generation of a computational model.

1. **Problem thinking:** Before committing to practice, it is worth thinking about the physics of the problem that is faced. In this stage, the analyst should consider the flow problem and try to understand as much as possible about it.
2. **Meshing:** In this stage, the analyst creates the problem domain shape using a CAD program. The domain is divided into cells or volumes using meshing, which can be done within the CFD package.

Defining Boundaries: The problem domain boundaries and their corresponding boundary conditions are established in the initial stage of a CFD simulation. This information, along with fluid parameters and physical properties, defines the specific flow problem to be solved. Advanced CFD software packages provide tools for grid generation, boundary definition, application of conditions, specification of initial conditions, setting fluid properties, and controlling numerical parameters.

Processing: Processing involves solving fluid flow equations using a computer. Meshing is followed by specifying input values and solving equations for each cell until convergence. This iterative process is the core of CFD software with limited visibility, but time-consuming.

Post-processing: The CFD post-processing program evaluates generated data, enabling numerical and graphical analysis of model results. It includes 2-D and 3-D visualizations like mesh sections, velocity vector plots, and scalar variable contour plots using color for differentiation. Results analysis ensures solution satisfaction and extraction of required flow data.

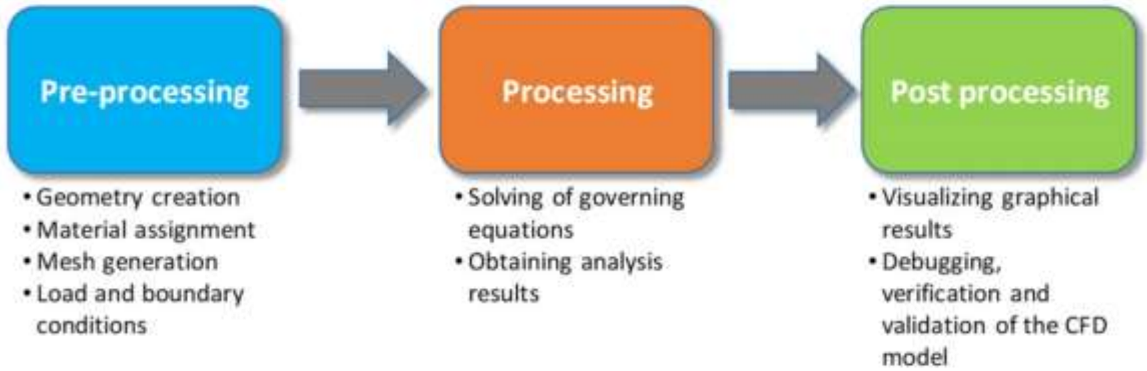

Figure 1: Stages in performing CFD analysis

Applications in simulations of transport phenomenon in the dairy processing:

Sterilization and pasteurization

Heat treatment equipment can be categorized into direct and indirect systems. Any heat treatment focuses on temperature, time, and the chemical and physical properties of the products, including the slowest heating zone. Ghani *et al.*, (2000) used CFD to simulate the process of sterilization. In the study, natural convection that occurs during sterilization in a pouch heated from all sides was simulated. The results showed different temperature profiles during sterilization process which highlight the migration of the slowest heating zone. Moreover, the dependency of concentration of alive bacteria on the temperature distribution was shown. The work assumes that the rate of bacterial inactivation is consistent with the first-order kinetics. The reaction rate constant is a function of temperature and is usually described by Arrhenius equation.

$$K_T = A \exp(-E/RT)$$

Accurate kinetic parameters such as reaction rate constants and activation energy were required to predict quality changes during food processing. Decimal reduction time is most commonly used. The relationship between the reaction rate constant and the decimal reduction time (D) is calculated according to following equation.

$$k_T = (2.303/D)$$

In the simulation, the value of the decimal reduction time at 121°C for Clostridium botulinum was 0.1 min, and the value of activation energy was 30×10^4 J mol^{-1}. The reaction rate constant k_T was calculated using above equation. Further, above equation is used to calculate the 'A' constant of Arrhenius equation ($A = 2.5 \times 10^{11}$–S^{-1}).

Anand Paul *et al.*, (2011) developed a CFD models for low and high temperature pasteurisation of canned milk for the first time and validated with the experiments. Moreover, the effect of can rotation (5, 50 and 100 rpm) on processing time and pasteurization value was investigated. A uniform heating (due to absence of slowest heating zone) of milk with reduced processing time was achieved in the rotated canned milk processing. Further, the study indicated that rotation process helps in maximum inactivation of microbes in shorter processing time.

Cooling and refrigeration

CFD simulations are extensively utilized for air flow calculations in designing refrigeration and freezing systems in the food industry. Temperature is a key parameter in determining the microclimate around food products. CFD enables optimization of cooling and freezing processes, including classic refrigeration devices and innovative freezing methods, to enhance efficiency, time, and energy utilization.

In ice cream manufacturing, freezing is a critical step that influences ice cream crystallization. Factors such as multiphase flow, ice crystal nucleation and growth, phase change, and viscous shearing impact the process. Miller *et al.*, (2011) conducted a study simulating ice crystallization in ice cream manufacturing. They focused on the dynamic freezing of sucrose solutions in a scraped-surface heat exchanger, examining the effects of multiphase phenomena, phase change, and shear on ice crystal nucleation and growth kinetics. Alvarenga *et al.*, (2021) utilized CFD simulations to analyze the maturation conditions of traditional sheep cheeses, specifically studying the effects of environmental factors such as humidity and temperature on the physicochemical and microbiological properties of the product.

Mixing

Mixing is a crucial process in dairy processing, ensuring homogeneity of different particles and components. CFD serves as a powerful tool for modeling mixing processes, including homogenization, solid mixing, and mixing of Newtonian and viscoelastic fluids. Additionally, CFD has been used in designing industrial mixers for fluid foods.

Homogenization

Homogenization is essential in food and dairy industries for creating emulsions and improving product quality. Understanding the valve's flow structure is crucial for optimizing homogenization devices in terms of energy efficiency and performance. CFD provides insights into turbulent structures in homogenizer valves, including velocity, pressure, and turbulent kinetic energy. Previous research used k- models with RANS equations for valve flow field analysis, yielding satisfactory results. However, limited studies focused on two-dimensional models with different k- models. Håkansson *et al.*, (2012) studied a three-dimensional valve and found inaccuracies near the gap exit and in reproducing turbulent kinetic energy at the gap entrance. The main understanding of flow structure in homogenizer valves are based on RANS approaches, exhibiting general information on the flow inside the valve with the time-averaging scheme. The "steady-state" RANS cannot reflect a detailed picture of instantaneous velocities or fluctuations that vanish due to the averaging process.

Mixing tank

Mixing tank is a device used for mixing one or more phases of matter with same or different physical and chemical characteristics, through the application of mechanical force. With the help of CFD, the phenomena in an agitated vessel can be predicted.

During mixing, a common method of enhancing the process is to use some kind of stirrer or paddle. CFD codes have been applied to optimize mixing by minimizing energy input and processing time. Previous research focused on energy distribution and mixing quality based on stirrer position, which was previously impossible to predict. CFD modeling of mixing in stirred tanks by Sahu et al. (1999) addressed impeller-vessel geometry, energy balance, and flow-field design linkage. The predicted mixing time values showed good agreement (within 5-10%) with published experimental data, using a 3D unsteady, pressure-based solver with a K-epsilon (RANS) turbulence model.

Designing of heat exchangers

CFD has found application in many areas of research of different types of heat exchangers: improper distribution of fluid flow, contamination, pressure drop and thermal analysis in the design and optimization phase. In simulations of processes with the use of heat exchangers, various turbulence models are used, e.g. standard k-ε, realizable k-ε, RNG k-ε and SST k-ω. Speed–pressure coupling schemes such as Semi-Implicit Method for Pressure Linked Equations (SIMPLE), Semi-Implicit Method for Pressure Linked Equations-Consistent (SIMPLEC), Pressure-Implicit with Splitting of Operators (PISO) are used to perform the simulation. In most cases, simulations give results in good agreement with experimental studies in the range of 2% to 10%.

The reliability of CFD results has reached the point where it has become an integral part of all design processes, leading to the elimination of the need for prototyping. The studies conducted by Piepiórka-Stepuk and Diakun (2014) shown that the design of the plate surface is a key issue in the development of heat exchangers. This work allowed simulating the distribution of fluid flow velocity in the channel between heat exchanger plates with different surface shapes. The results showed that the velocity field and streamlines favour the use of panels with a corrugated surface. The velocity distribution in the duct between the heat exchanger plates is an important parameter of the cleaning efficiency of a damaged system.

Spray Drying

CFD simulations are crucial for understanding and enhancing the spray-drying process, as obtaining measurements of air flow, temperature, particle size and humidity in large-scale dryers is challenging and costly. Advanced techniques like particle image velocimetry (PIV) and temperature/humidity sensors can complement CFD simulations to provide detailed insights into the flow pattern and conditions inside the drying chamber. Submodels in CFD simulation include particle tracking, droplet drying (heat and mass transfer), and particle quality. Accurate heat dose selection and process optimization are crucial for nutritional value preservation and avoiding harmful compound formation. Optimization using CFD offers economic benefits by reducing energy consumption (Kuriakose & Anandharamakrishnan, 2010).

(i) Drying fluid flow simulation in the drying chamber

The air-flow pattern in the chamber affects the movement of the particles, which subsequently effects the residence time of the particles, drying rates (i.e. heat and mass transfer) and whether the particles are deposited on the chamber wall or escape through the outlet pipe. Hence, getting an accurate prediction of the flow field in the drying chamber is a prerequisite for the subsequent modelling effort. Most spray dryer CFD simulations are performed using commercial codes such as FLUENT, CFX and STAR-CD software tools.

Turbulence modeling

Selecting an appropriate turbulence model is crucial for spray-drying simulation, considering accuracy and computational requirements. The standard k-ε model is commonly used due to its robustness, low computational demand, and reasonable accuracy. While effective for simple flows, it struggles to capture complex flow patterns, like swirling flows in counter-current spray dryers, where swirling effects are not considered. Thus, an optimal turbulence model should be chosen to accurately simulate fluid flow in spray dryers.

Bayly *et al.,* (2004) reported that the RSM can give better results for turbulent swirling flows in a counter-current spray dryer. The shortcoming of the standard $k - \varepsilon$ model is attributed to its assumption of isotropic turbulence. Other $k - \varepsilon$ related models, such as the renormalized $k - \varepsilon$ (RNG) and realizable $k - \varepsilon$ (RKE), accommodate the swirling flow model and mathematical function to ensure positivity in turbulence stresses.

Chamber design and flow configuration

The effect of different chamber geometries (cylinder-on cone, lantern, hour-glass and pure cone) on the drying performance and the particle residence time was studied by Huang *et al.,* (2003). They suggested that it is possible to change the chamber geometry for better utilization of the dryer volume. The use of CFD to obtain uniform flow distribution in a plenum chamber with a single, off-axis, inlet pipe was studied by Southwell *et al.,* (2000). The flow distribution was investigated and the result was confirmed by experimental data for selected configurations. They concluded that CFD can be used for several design alternatives for overcoming poor flow distribution. Apart from the conventional vertical design of the spray dryers, the use of horizontal design is gaining importance nowadays. Studies revealed that use of good chamber design and incorporation of fluidized bed will improve the drying performance. In spray drying, the majority of works of CFD simulations have been performed on co-current flow.

Conclusion

CFD is a powerful tool used in the food industry for understanding physical phenomena in 2D and 3D geometries. It has wide applications in the dairy industry, from processing to packaging. CFD simulations optimize flows, temperature distribution, and mechanical changes, leading to process optimization and cost reduction. In the future, CFD will be indispensable for enhancing process efficiency in the dairy and food industry.

References

Alvarenga, N., Martins, J., Caeiro, J., Garcia, J., Pássaro, J., Coelho, L., & Dias, J. (2021). Applying computational fluid dynamics in the development of smart ripening rooms for traditional cheeses. Foods, 10(8), 1716.

Bayly, A. E., Jukes, P., Groombridge, M., & McNally, C. (2004, August). Airflow patterns in a counter-current spray drying tower-simulation and measurement. In Proceedings of the 14th International Drying Symposium (pp. 775-781).

Ghani, A. A., Farid, M. M., & Chen, X. D. (2002). Numerical simulation of transient temperature and velocity profiles in a horizontal can during sterilization using computational fluid dynamics. Journal of Food Engineering, 51(1), 77-83.

Håkansson, A., Fuchs, L., Innings, F., Revstedt, J., Trägårdh, C., & Bergenståhl, B. (2012). Experimental validation of k–ε RANS-CFD on a high-pressure homogenizer valve. Chemical engineering science, 71, 264-273.

Kuriakose, R., & Anandharamakrishnan, C. (2010). Computational fluid dynamics (CFD) applications in spray drying of food products. Trends in Food Science & Technology, 21(8), 383-398.

Miller, M. J., Xin, X. J., Pei, Z. J., & Schmidt, K. A. (2011). Ice Crystallization in Ice Cream Manufacturing by Coupled Computational Fluid Dynamics and Population Balance Method. In International Manufacturing Science and Engineering Conference (Vol. 49460, pp. 203-208).

Piepiórka-Stepuk, J., & Diakun, J. (2014). Numerical analysis of fluid flow velocity between plates channel of heat exchanger by different surface configuration in reference to the effects of cleaning. Italian Journal of Food Science, 26(2), 210.

* * * * * * * *