Journal of Food Engineering 123 (2014) 8–16

Contents lists available at ScienceDirect

Journal of Food Engineering

journal homepage: www.elsevier.com/locate/jfoodeng

Effects of mesh refinement, time step size and numerical scheme on the computational modeling of temperature evolution during natural-convection heating

Ziynet Boz [a], Ferruh Erdogdu [a,*], Mustafa Tutar [b,c]

[a] Department of Food Engineering, University of Mersin, 33343 Ciftlikkoy-Mersin, Turkey
[b] Mechanical and Manufacturing Department, MGEP Mondragon Goi Eskola Politeknikoa, Loramendi 4 Apartado 23, 20500 Mondragon, Spain
[c] IKERBASQUE, Basque Foundation for Science, 48011 Bilbao, Spain

ARTICLE INFO

Article history:
Received 25 February 2013
Received in revised form 25 July 2013
Accepted 2 September 2013
Available online 14 September 2013

Keywords:
CFD
Mesh refinement
Time step size
Numerical solver

ABSTRACT

A computational fluid dynamics (CFD) analysis includes creating geometry, dividing it into elements, discretizing governing equations and solving with suitable numerical schemes. Refining mesh and lowering time step size increase accuracy. However, this may not always give the best result since convergence depends upon parameters of time step size, mesh resolution and numerical scheme. Therefore, the objective of this study was to determine the effects of these on the accuracy of a CFD solution. For this purpose, canning example was used, and results obtained with ANSYS CFX (v12.1.) were experimentally validated. Effects of time step size, mesh resolution and numerical scheme on temperature and velocity field evolution were determined. Mesh refinement affected temperature distribution significantly and surprisingly led to less accuracy. The reasons for this behavior were presented and explained comprehensively.

1. Introduction

Thermal processing provides safety and stability to food products until their consumption. Controlling adequacy and efficiency of thermal processing by computational fluid dynamics (CFD) methodology has recently been used to lead designing processes and/or optimizing existing processes for higher quality products (Verboven et al., 2004). CFD methodology uses numerical methods to solve partial differential forms of governing equations of continuity, heat and momentum. A typical CFD analysis consists of creating geometry, dividing it into small elements (mesh) to form computational geometry, discretizing governing equations and solving with suitable numerical schemes (Lomax et al., 2001). A variable to determine, such as temperature at a certain point in the computational domain, is predicted by using iterative solutions until the solution reaches a pre-set convergence criterion through the given time step size of Δt (from t to $t + \Delta t$; Cornelissen et al., 2007). Hence, all modeling parameters, i.e. mesh size, time step size and convergence criteria, qualify the success of a proper CFD solution. Accuracy of CFD solutions is generally improved with increased number of elements (Tu et al., 2008; Sorensen and Nielsen, 2003). As a general rule, increasing the number of elements within a computational geometry is carried out until a mesh independent solution is achieved (Sorensen and Nielsen, 2003; Wang and Zhai, 2012).This should eventually provide minimum difference between the solutions. However, this might lead to an undesired increase in the computational time (Norton and Sun, 2006). Due to the computational power limitations, increasing the number of mesh elements in the computational geometry may not always be attainable, and use of coarser meshes might still give accurate results opposite to the theory of that *the finer the mesh resolution the higher the accuracy*.

Hoang et al. (2000) investigated the airflow in a cold store and used coarser mesh distribution to obtain velocity field distribution considering the computational power. Gas circulation inside of a cheese ripening room was modeled by Mirade and Daudin (2006), and some inaccuracies in the magnitude of airflow were linked to the possible unachieved mesh independency due to the computational memory restrictions. However, in some cases, exceptions are observed in validation and accuracy tests, and more accurate results are obtained via the coarser mesh structures. Harral and Boon (1997) performed a CFD study to determine airflow velocity distribution in a ventilated livestock. Even though the mesh independency was obtained with the finest mesh in most cases, it was the coarsest mesh structure that best fitted to the experimentally measured velocities in one of the cases. This was defined as "coincidental" in this study. Salim and Cheah, 2009 investigated CFD parameters of the wall-bounded turbulent flows

* Corresponding author. Tel.: +90 533 812 0686; fax: +90 324 361 0032.
E-mail addresses: ferruherdogdu@yahoo.com, ferruherdogdu@mersin.edu.tr (F. Erdogdu).

0260-8774/$ - see front matter © 2013 Elsevier Ltd. All rights reserved.
http://dx.doi.org/10.1016/j.jfoodeng.2013.09.008

Nomenclature

c_p	fluid specific heat (J/kg K)
Co	Courant number
g	gravitational acceleration (m/s^2)
k	fluid thermal conductivity (W/m K)
N	number of data points
P	pressure (Pa)
$RMSE$	root mean square error (°C)
t	time (s)
T	temperature (K)
T_{ref}	temperature (K)
v_r	velocity component in r-direction (m/s)
v_θ	velocity component in θ-direction (m/s)
Greek symbols	
β	thermal expansion coefficient (1/K)
μ	dynamic viscosity (Pa s)
ρ	density (kg/m^3)
ρ_{ref}	reference density (kg/m^3)

in terms of near-wall treatment. It was demonstrated that the lowest resolution of mesh distribution gave the best accuracy when compared to the experimental results. Druzeta et al. (2009) made a mesh refinement study in a two-dimensional shallow water model and concluded that the mesh refinement adversely affected the accuracy of the results leading to a doubt on a generally accepted rule of "*the finer the mesh resolution the higher the accuracy*".

Convergence of solutions depends upon various parameters such as applied time step size, mesh resolution and numerical scheme to relate boundary conditions with computational elements. Food engineering literature generally makes use of first order hybrid and upwind numerical schemes due to their higher convergence features, but mesh refinement does not always serve the purpose of gaining the desired accuracy (Norton and Sun, 2006). Based on these, the objectives of this study were to determine the effects of mesh refinement, applied numerical schemes and time step size on the accuracy of a CFD solution. For this purpose, thermal processing example of canned water was used, and computational results, obtained with various numerical schemes, mesh structures and applied time step sizes, were validated with experimental results.

2. Materials and methods

For the stated objectives, the study was completed in two parts. In the experimental part, thermal processing of a canned water example was carried out. After experimental temperature measurements were performed, initial and boundary conditions were specified, CFD simulations were completed using Ansys CFX 12.1 (Ansys, Inc., Canonsburg, PA), and effects of mesh refinement, applied time step size and numerical scheme were determined.

2.1. Experimental methodology

Experiments were completed to determine initial and boundary conditions and validate the CFD solution properly. For temperature measurements, distilled water filled cylindrical cans (73 mm in diameter and 110 mm in height) was used. Type-T needle thermocouples (Ecklund-Harrison Technologies, Fort Myers, FL) were placed tightly using joint gaskets and locking receptacles to measure temperature change during heating. Headspace in the can was less than 5% of the total volume to avoid its possible effects on heat transfer. Water filled cans were then sealed imperviously with a sealing machine (MAC-230, Umar Makina Sanayii, Istanbul, Turkey) and located horizontally in a vertical retort (OMS Lab 20, Osmanli Makina, Balikesir, Turkey). Variations in temperature changes at three different locations of the cans to validate with the simulation results were recorded with a data acquisition system (Keithley Instruments, Cleveland, OH) under boiling conditions. For each case, different cans were used, and ten experiments were carried out. The results were reported to be the averages and standard deviations. Fig. 1 shows the locations where temperature changes were recorded in the can during the various experiments. The RMSE (root mean square error) statistical values (Atkinson, 1993) were used to compare experimental measurements and simulation results:

$$RMSE = \sqrt{\frac{1}{N}\sum_{i=1}^{N}(T_{\text{experimental}} - T_{\text{simulations}})^2} \quad (1)$$

where N is the number of experimental data and T is the temperature values obtained from experimental measurements and simulations at the center point of the can. The lower RMSE value demonstrated the better the compatibility of the simulation results.

2.2. CFD simulations

2.2.1. Computational geometry

Computational model was a single phase (water) model. Even though a certain headspace was left in the cans (<5%) to prevent the cans from bulging during heating, this was not accounted in the simulations. Including the headspace in the model would

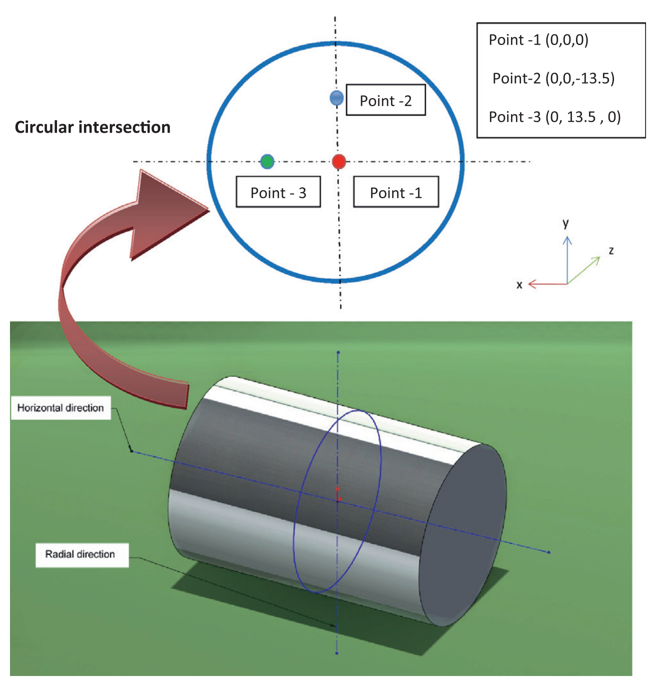

Fig. 1. Locations where temperature changes were recorded experimentally.