Journal of Microwave Power and Electromagnetic Energy, 44 (4), 2010, pp. 187-197
A Publication of the International Microwave Power Institute

Numerical Modeling of Continuous Flow Microwave Heating: A Critical Comparison of COMSOL and ANSYS

D. Salvi[1,2], Dorin Boldor[1], J. Ortego[1], G. M. Aita[2], C. M. Sabliov[1]

[1] Department of Biological and Agricultural Engineering, 149 E.B. Doran Bldg.
Louisiana State University Agricultural Center
Baton Rouge, LA 70803

[2] Audubon Sugar Institute
Louisiana State University Agricultural Center
3845 Hwy 75, St. Gabriel, LA 70776

Received: December 20, 2009
Accepted: December 11, 2010

ABSTRACT

Numerical models were developed to simulate temperature profiles in Newtonian fluids during continuous flow microwave heating by one way coupling electromagnetism, fluid flow, and heat transport in ANSYS 8.0 and COMSOL Multiphysics v3.4. Comparison of the results from the COMSOL model with the results from a pre-developed and validated ANSYS model ensured accuracy of the COMSOL model. Prediction of power loss by both models was in close agreement (5-13% variation) and the predicted temperature profiles were similar. COMSOL provided a flexible model setup whereas ANSYS required coupling incompatible elements to transfer load between electromagnetic, fluid flow, and heat transport modules. Overall, both software packages provided the ability to solve multiphysics phenomena accurately.

KEYWORDS: Microwave processing, numerical modeling, COMSOL Multiphysics, ANSYS Multiphysics, resonant cavity

INTRODUCTION

In continuous flow microwave heating of liquids, volumetric heat generated as a function of electric field distribution and dielectric properties of the fluid is transferred in the flowing fluid by conduction and convection. Comprehensive 3-D numerical modeling of the process requires coupling of energy and momentum equations to Maxwell's electromagnetic equations. [Le Bail et al. 2000] predicted temperature profiles in fluids under continuous microwave heating assuming uniform volumetric power in the core of the flow. [Ratanadecho et al. 2002] used a 2-D finite difference time domain (FDTD) formulation to solve Maxwell's equations within a finite control volume based on the SIMPLE algorithm to solve heat transport and fluid flow equations in rectangular duct geometry; an iterative computational scheme was used to couple Maxwell's equations with momentum and heat transfer equations. [Zhu et al. 2007a, 2007b, and 2007c] used a similar algorithm for numerical modeling of continuous flow microwave heating in cylindrical and rectangular ducts.