Brief Report
Volume 13:6, 2022

Global Journal of Technology and Optimization

ISSN: 2229-8711

Open Access

Thermoelectric System Design and Evaluation Using Computational Fluid Dynamics

Rizzi Zhung*

Department of Electrical and Computer Engineering Biomedical Engineering, Nanyang Technological University, Nanyang Ave, Singapore

Introduction

Due to the availability of inexpensive computer power and sophisticated modelling tools like computational fluid dynamics, the design of thermal processes in the food industry has experienced significant advancements in the recent two decades (CFD).CFD employs numerical techniques to resolve the complicated dynamics governing many food-processing systems by solving the non-linear partial differential equations of fluid mechanics and heat transfer. To develop solutions with a high level of physical realism without the significant costs associated with experimental investigations, CFD can be utilised in thermal processing applications to build three-dimensional models that are both spatially and temporally representational of a physical system. As a result, CFD is becoming more and more important in the development of conventional process optimization in the food sector. The last few of decades have seen a notable advancement in the application of fluid motion and heat transfer principles to design issues in the food sector. Previously requiring either extremely expensive experimental rigs or too simplistic computations, problems including heat and mass transfer, phase change, chemical processes, and complicated geometry can now be represented with a high level of spatial and temporal accuracy on home computers. Heat transport to and from meals can trigger intricate chemical and physical changes in them due to the complicated thermo-physical characteristics of food [1.2].

Description

To discretize the transport equations, CFD code developers can choose from a wide variety of numerical approaches. The three most significant ones are the ones dealing with finite difference, finite elements, and finite volume. The oldest method in use is the finite difference method, and there are numerous instances of it being employed in the food sector. Finite difference is not, however, used commercially since it is difficult to deal with uneven geometry. Additionally, the development of unstructured meshing technology that can manage the intricate three-dimensional geometries encountered in industry is a goal of the current commercial CFD coding trend. Finite difference appears to have little potential for use in industrial CFD applications. The integral transport equations that drive the physical process are expressed using finite volume approaches in conservation form, and the volume integrals are then transformed into surface integrals by means of Gauss's divergence theorem. It is simple to interpret since it is a direct extension of the control volume analysis, which many engineers utilise in thermodynamics and heat transfer applications, among other things. In order to achieve a systematic description of the changes in mass, momentum, and energy as fluid crosses the boundaries of discrete spatial volumes inside the computational domain, it is therefore physically intuitive to describe the equation system using finite volumes. Additionally, algebraic equations produced by finite volume approaches support solver resilience [3,4].

A simulation's volume mesh is a mathematical description of the area or geometry that needs to be addressed. Tetrahedral, hexahedral hybrid, and even polyhedral meshes can now be included into commercial software, which is one of the most significant developments in meshing technology in recent years. This has improved the ability to find solutions for many industrial applications by enabling mesh to be fitted to any arbitrary geometry. Additionally, when it comes to characterising the physics of the problem, some contemporary commercial CFD codes encourage relatively little interaction between the user and specific mesh pieces. The user is able to focus more on the specifics of the geometry and the transfer of simulation attributes because to this decoupling of the physics from the mesh. The approximation assumes that the buoyancy element of the momentum equations is the only place where the density differentials of the flow are necessary. Additionally, it is believed that temperature and density have a linear relationship and that all other significant fluid parameters are constant. The density variation for a multi-component fluid medium can also be determined, but this connection only takes into account a single component fluid medium. Unfortunately, with significant temperature differences, the Boussinesq approximation is not accurate enough. As a result, in these situations, another technique for linking the temperature and velocity fields is required [5].

Due to the presence of high flow rates and heat transfer interactions, thermal processes are typically characterised by turbulent motion. For laminar flows, the Navier-Stokes equations can currently be solved directly. There are many turbulence models for turbulent flow regimes that are effective in various applications. However, it should be highlighted that none of the current turbulence models are comprehensive, meaning that their ability to forecast outcomes depends heavily on the characteristics of turbulent flow and its geometry. Three transport equations each for the turbulent fluxes of each scalar property and one transport equation for the rate of turbulence energy dissipation make up the Reynolds stress closure model (RSM), which typically consists of six transport equations for the Reynolds stresses [2,4].

Conclusion

For more than a decade, CFD has actively contributed to the design of thermal processes. Since application-specific models may now be easily added to the software via user-defined files, simulations have become more sophisticated in recent years. The analysed research demonstrate the significance of maintaining a high level of accuracy through prudent decisions made during model construction, since several studies provide in-depth validation activities. It is feasible that CFD will continue to offer answers for transport phenomena as computing power continues to advance unceasingly, leading to better design of thermal processes in the food business.

Acknowledgement

None.

Conflict of Interest

The authors reported no potential conflict of interest.

***Address for Correspondence:** Rizzi Zhung, Department of Electrical and Computer Engineering Biomedical Engineering, Nanyang Technological University, Nanyang Ave, Singapore; E-mail: zhung.rizzi @up.ac.za*

Date of Submission: 03 June, 2022, Manuscript No.GJTO-22-71121; **Editor Assigned:** 05 June, 2022, PreQC No. P-71121; **Reviewed:** 10 June, 2022, QC No. Q-71121; **Revised:** 15 June, 2022, Manuscript No. R-71121; **Published:** 20 June, 2022, DOI: 10.37421/2229-8711.2022.13.299

References

1. Norton, Tomás, Brijesh Tiwari and Da-Wen Sun. "Computational fluid dynamics in the design and analysis of thermal processes: A review of recent advances." *Critical Rev Food Sci Nut* 53 (2013): 251-275.

2. Da Ronch, A., M. Ghoreyshi and K.J. Badcock. "On the generation of flight dynamics aerodynamic tables by computational fluid dynamics." *Prog Aerospace Sci* 47 (2011): 597-620.

3. Fidkowski, Krzysztof J and David L. Darmofal. "Review of output-based error estimation and mesh adaptation in computational fluid dynamics." *AIAA* 49 (2011): 673-694.

4. Niemeyer, Kyle E and Chih-Jen Sung. "Recent progress and challenges in exploiting graphics processors in computational fluid dynamics." *J Supercom* 67 (2014): 528-564.

5. Rizzi, Arthur and Björn Engquist. "Selected topics in the theory and practice of computational fluid dynamics." *J Com Phys* 72 (1987): 1-69.

How to cite this article: Zhung, Rizzi. "Thermoelectric System Design and Evaluation Using Computational Fluid Dynamics." Glob J Tech Optim 13 (2022): 299.