Simulating The Cooling Of Medical Ct-Scanners: Part 2: Results

Mariam K. Hafshejani[1], Ali Darasaraei[2], Fakhrodin Dadjoo[2], Fakhrodin Alimoradi[2], Ali Falavand[3], Armin Arad[4*]

[1]Shahrekord University of Medical Sciences, Shahrekord, Iran
[2]Civil Engineering Group, Chaloos Branch, Islamic Azad University, Chaloos, Iran
[3]Engineering Group, Ahvaz Branch, Islamic Azad University, Ahvaz, Iran
[4] North Khorasan University of Medical Sciences, Bojnurd, Iran
Corresponding author email: aarad1384@yahoo.com

Abstract: CT-scanning is used as an non-invasive detecting method in medical applications. The previous paper on this topic is dedicated to express the numerical method applied for simulating cooling system of CT-scanner system used in medical applications. In Part I of this paper we study the convection/diffusion mechanism of heat transfer during passing a cold fluid from the top of a hot plate of CT-scanner. This paper is concerned with the analysis of the numerical results achieved from the developing code. For verification, the approach to determining an adequately fine mesh is to perform exploratory simulations for different mesh sizes. Results obtained from this simulation describe the convective and diffusive rate of heat transfer for such CT-scanner cooling systems
[Hafshejani M K, Darasaraei A, Dadjoo F, Alimoradi F, Falavand A, Arad A. **Simulating the Cooling of Medical CT-Scanners: Part 2: Results.** *Life Sci J* 2012;9(2):1311-1315] (ISSN:1097-8135). http://www.lifesciencesite.com. 195

Keywords: CT-scanner, CFD simulation, Heat transfer

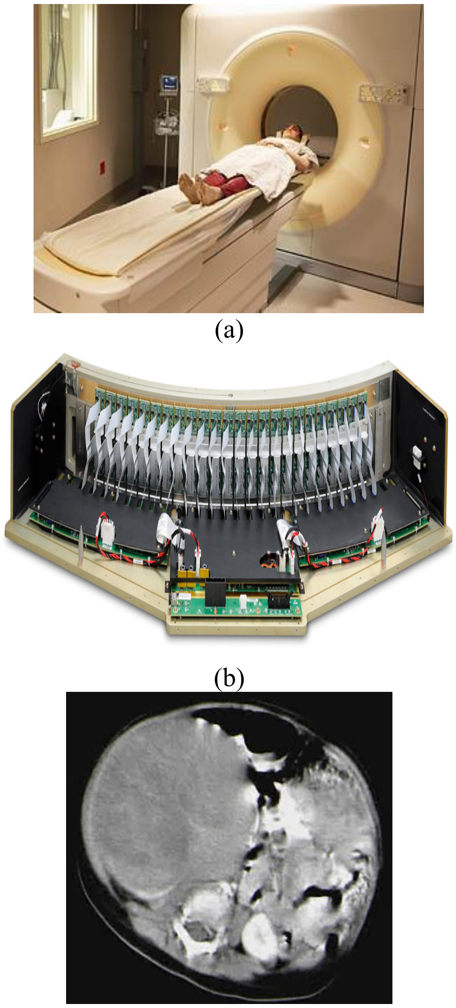
1. Introduction

In recent decades computed tomography (CT) scanners are used in medical applications and detecting disease (S. Hassan and G. Hassan, 2011). These devices can create 2-D images from patient body structures which help doctors to detect cancers, arteries etc.. Figure 1(a-c) shows the CT imaging system, the gantry and a typical produced image. As seen in figure 1(a), computed tomography imaging system primarily includes the gantry and patient table. The gantry (figure 1(b)) is a moveable frame that contains the imaging system and bears high substantial amount of heat. It is well established that the x-ray tube of CT-scanner generates large amount of heat which may causes to failing the system if no appropriate cooling has been conducted. The cooling system must transfer high heat levels generated during the high speed rotation of the anode and the bombardment of electrons upon the anode surface (Reddinger, 1997; Ahmadi and Marghmaleki, 2011). The x-ray tubes heat capacity is expressed in heat units. Modern CT scanners bear a heat capacity of 3.5-5 million heat units (Reddinger, 1997). Many CT x-ray tubes utilize a combination of oil and air cooling systems to eliminate heat and maintain continuous operational capabilities (Reddinger, 1997).

Originally, CFD was born out of the need to use in medical applications to analyze airplane components (www.ehow.com; Biao *et al.* 2012). This occurred during the 1930s, but CFD as a branch of science did not begin to take off until the 1960s due to the improvement of computing power. CFD is now studied at many major universities and is applied to many fields of science (www.ehow.com). In this work CFD is used for simulating the heat transfer in CT gantry surface via fluidic cooling system. Determining the temperature distribution in the CT gantry model is the main aspect of any heat analysis process. Technically, heat transfer analysis is important and uses numerical calculations of heat transfer coefficients for determining boundary conditions of substances (Kovtanyuk *et al.*, 2012; Wu *et al.*, 2011). Irrespective of the complexity of the problem, CFD heat transfer analysis allows engineers to calculate fluid flow properties and the effects of temperature sensitive material, non-linearity, and fluid-surface contact condition on the mechanical component. All CFD analysis and simulations are separated into three distinct parts: pre-processing, processing, and post-processing (Norton *et al.*, 2017) .

In the pre-processing phase, a problem is defined. This problem is the "what" of what is to be studied. Once the problem is defined, conditions must be defined for the problem. These are known as initial conditions. Then, boundary conditions must be defined to act as boundaries for the problem. In this phase, it is also decided how this problem will be studied, such as what method of discretization, what numerical methods to use, and what programming language to use.

In the processing phase, the computer code used to solve the problem at hand has been written and is compiled. This phase is mainly user-free because the computer is performing hundreds and thousands of calculations in order to simulate the problem at every step in time. The end result of this phase is a large collection of data.



(a)

(b)

(c)

Figure 1. (a) the CT-scanner (b) CT gantry anode and (c) a typical CT image.

In the post-processing phase, many thousands of calculations have been performed and data relevant to the study has been produced. This data is then filtered and converted into meaningful data.

The previous paper of this work is mostly with the per-processing stage of the performed CFD work for analyzing CT scanner cooling system. In this paper, a comprehensive analysis of the results as the stage of "pots-processing" in our CFD work is presented and discussed for the CT-scanner cooling system model.

2. Boundary conditions

There are several boundary conditions such as the inflow velocity profile, constant temperature, constant heat flux and symmetry conditions. The figure below shows the boundary conditions we considered in this problem. The boundary conditions applied for this problem is shown schematically in Figure 2. The values of L, W, T up, T left, T∞, h can be considered as input data.

3. Results

Available results can very well demonstrate dynamic variation of temperature profile with time. As seen, at the beginning, the temperature contours show the heat transfer diffusion through the bulk of the fluid. However, after passing a certain time, the impact of convective heat transfer becomes significant. The following figures (figure 3-8) presents the temperature contours through the bulk of the fluid. It should be noted that the mesh size has been reduced so that no change in temperature on each grade is demonstrated.

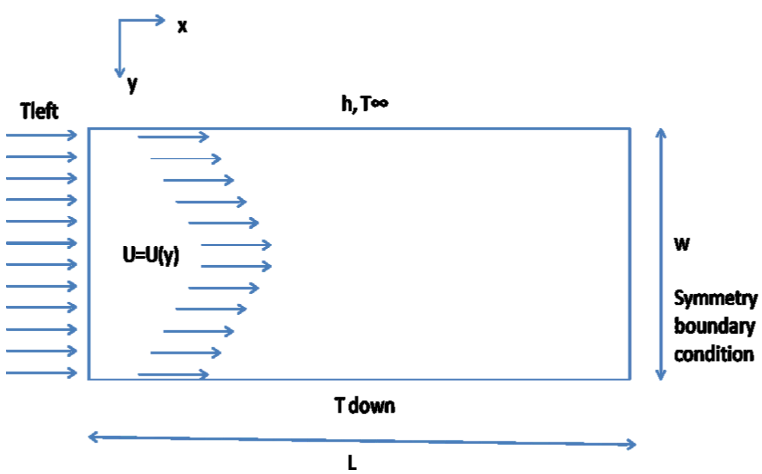


Figure 2. Boundary conditions

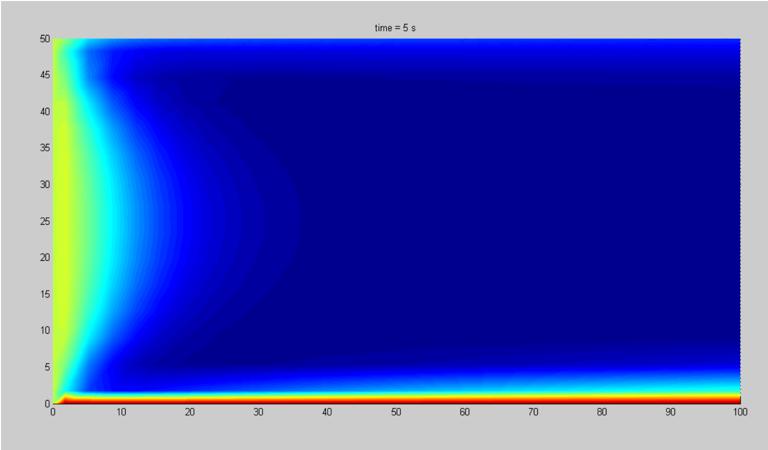


Figure 3. Temperature contour at the time 5s after the beginning of the flow

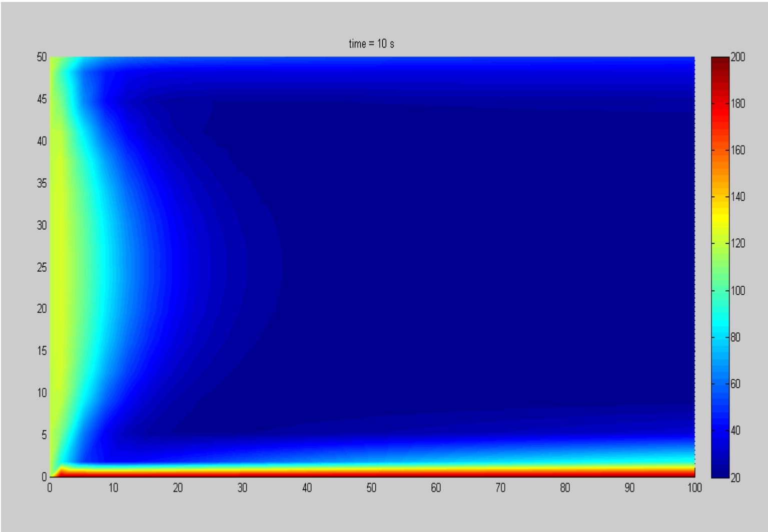


Figure 4. Temperature contour at the time 10s after the beginning of the flow

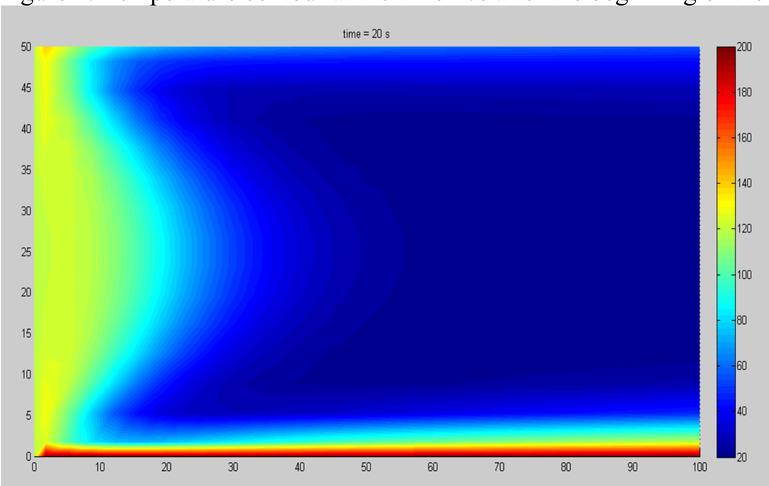


Figure 5. Temperature contour at the time 20s after the beginning of the flow

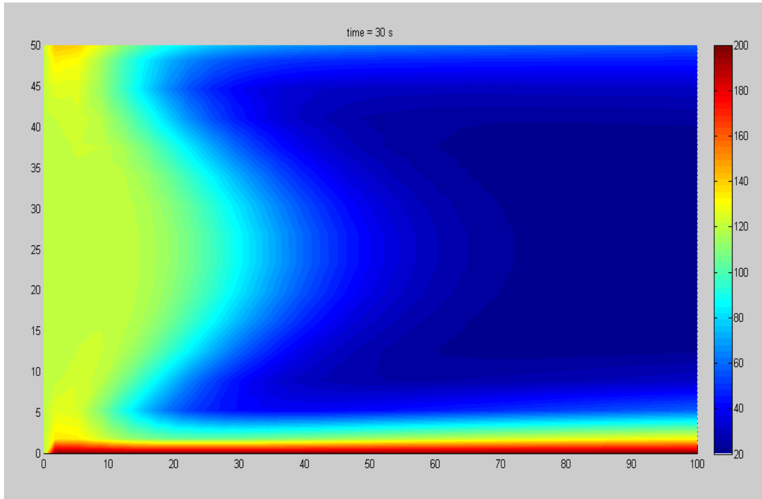


Figure 6. Temperature contour at the time 30s after the beginning of the flow

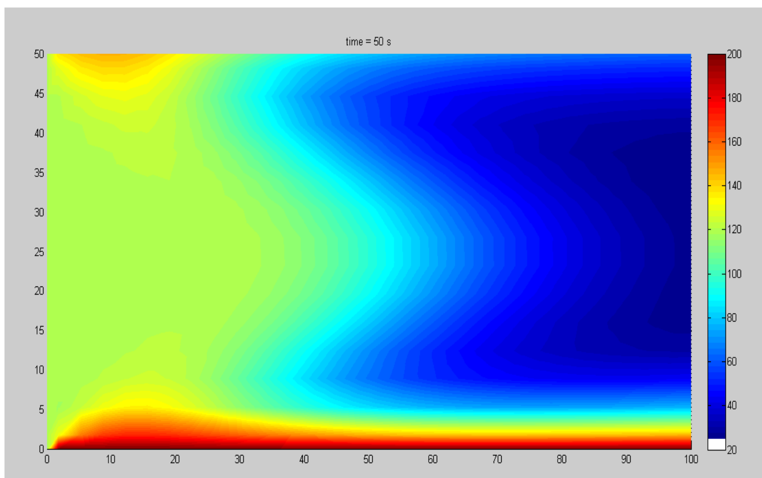


Figure 7. Temperature contour at the time 50s after the beginning of the flow

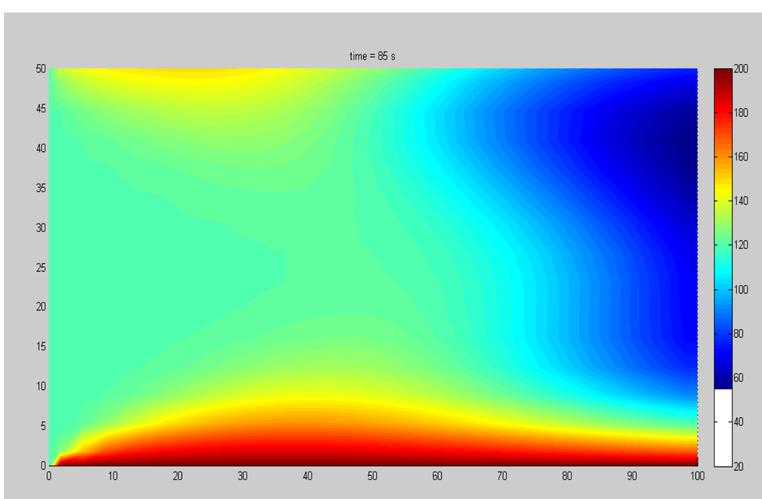


Figure 8. Temperature contour at the time 85s after the beginning of the flow

4. Conclusion

CT-scanners are used as an powerful detecting radiological devices in medical applications. The Part I of this paper we express the numerical method applied for simulating cooling system of CT-scanner system used in medical applications. This work is mostly dedicated to the study of heat transfer mechanism through the bulk of a fluid, passing above a hot plat of a CT-scanner gantry. The work is performed as a verification step in the development of our CFD code, performed based on the Finite Volume Method, to handle heat transfer problems in fluids. The obtained temperature contours very well describe the mechanism of heat transfer across the bulk of the fluid, affected by the hydrodynamic characteristics of the problem. Results show the variation of parameters during the cooling of CT-scanner.

Acknowledgement

The author would like to thank all the people who contribute this project

Corresponding Author:
Armin Arad
North Khorasan University of Medical Sciences
Bojnurd, Iran
E-mail: aarad1384@yahoo.com

References

1. Ahmadi M, Marghmaleki I S. Numerical simulation of turbulent flow in channels with three-dimensional blocks; Life Science Journal, 2011; 8(4): 511-516.
2. Biao H, Wei W , Xixu W, Jue W, Meng Y. Numerical Simulation to Get Flow Pattern in Modified Carotid Artery Bifurcation Model Using PIV; Life Science Journal, 2012; 9(3): 1296-1301.
3. Hassan M S, Hassan M G. Role of Multislice CT in Assessment of Carotid Stenosis; Life Science Journal, 2011; 8(4): 753-756
4. http://www.ehow.com.
5. Kovtanyuk A E, Botkin N D, Hoffmann K H. Numerical simulations of a coupled radiative-conductive heat transfer model using a modified Monte Carlo method; International Journal of Heat and Mass Transfer 2012; 55(4): 649-654.
6. Norton T, Sun D W, Grant J, Fallon R, Dodd V. Applications of computational fluid dynamics (CFD) in the modelling and design of ventilation systems in the agricultural industry: A review.; Bioresource Technology 2007; 98(12): 2386-2414.
7. Reddinger W L. CT Instrumentation & Physics, OutSource Inc., 1997.
8. Wu Z, Caliot C, Flamant G, Wang Z. Numerical simulation of convective heat transfer between air flow and ceramic foams to optimise volumetric solar air receiver performances; International Journal of Heat and Mass Transfer 2011; 54(7–8):1527-1537.

5/5/2012