American Scientific Research Journal for Engineering, Technology, and Sciences (ASRJETS)

ISSN (Print) 2313-4410, ISSN (Online) 2313-4402



http://asrjetsjournal.org/

# A Numerical Investigation of Heat Transfer in a Rotating Enclosure

Johnny Issa[a]*, Najib Saliba[b], Ghady Dib[c]

[a,b,c] *Faculty of Engineering, University of Balamand, Al Kurah, Lebanon*

[a]*Email: johnny.issa@balamand.edu.lb*

[b]*Email: najib.saliba@balamand.edu.lb*

[c]*Email: deebghady@gmail.com*

**Abstract**

Many thermal techniques are used in various industries in order to enhance cooling and heating of fluids in contact with a surface. Some of these techniques include rotation of fluids inside an enclosure in an attempt to improve the thermal exchange between the fluid flows and the contact surfaces. These methods involve complex heat transfer mechanisms, therefore the need of numerical simulations is crucial for understanding them. In the current study, a numerical investigation of the fluid flow and heat transfer in a rotating enclosure was completed. First, heat transfer from a fluid in a cylindrical enclosure with forced convection boundary conditions was numerically simulated and compared to known analytical solutions for benchmarking purposes. An excellent agreement between the numerical and analytical solutions was obtained. Then, heat transfer and internal flow behaviors in a container rotating at different angular velocities were numerically studied. Velocity and temperature distributions were studied. Comparison between numerical and analytical solutions was completed for cases where analytical solutions are available. For low Reynolds number, temperature distribution in a full container was observed not to depend on the rotation rate which is in good agreement with the analytical findings from the non-dimensional energy equation.

***Keywords:*** computational methods; fluid mechanics; heat transfer; rotating enclosure.

------------------------------------------------------------------------

* Corresponding author.

## 1. Introduction

Many thermal techniques consist of heating or cooling of fluids in an enclosure for a determined time and temperature. Time is very important to control through the thermal process since many variables such rotational speed, container location, mode of rotation, headspace and product properties, affect the heat transfer rate.

Abdul Ghani et al. investigated the rotation effects on purification method of a viscous fluid in a horizontal container, using numerical analysis. Based on solving the governing equations of mass, momentum and energy conservation for the three-dimensional horizontal container, their target was to present the transient temperature distribution caused by natural and forced convection heating and to compare it with those for a non-rotating container. As a result of the numerical simulations, the slowest heating zone (SHZ) was found to be reduced to 5% of the total volume of the rotated container [1].

Erdogdu et al. analyzed the effect of rotation on heat transfer process in rotating container containing fluids with headspace. The objective of their study was to analyze the evolution of the temperature distribution in cans containing headspace with the fluids while rotating, in order to verify the function of headspace in increasing the convective heat transfer rate. The optimal rotating process conditions were found to depend on the fluid physical properties (viscosity and density) and rotation speed. In fact, viscosity is the major factor that affects the heat transfer rate in this case. [2].

Using CFD simulations, Kannan et al. studied heat transfer from a fluid in an enclosure in a still retort which it led them to predict the heat transfer coefficient. Also, they compared heat transfer coefficient with respect to various surfaces of the uniformly heated cylindrical container. They found that the curved surface and due to its large surface area, dominates the heat transfer process [3]. The objective of the current study is to numerically investigate rotation effects on enhancing the heat transfer from/to a fluid in an enclosure. The influence of varying the wall boundary conditions and Reynolds number on the velocity and temperature distribution with respect to time was also investigated in this study.

## 2. Validation Problem

Beverage industries use thermal processing method to cool liquid inside storage cans in a faster way without distorting the beverage taste like the traditional way. In fact they introduced rotation process to the problem in order to increase the heat transfer.

In this problem, heat transfer from a stationary fluid in a cylindrical container is used as a validation problem. The application of a higher heat transfer coefficient by using water as the working fluid and forced convection method is studied.

### *2.1. Anlytical Study*

Analytical study was completed where the heat diffusion equation was solved in time and space, and the solution is as follows:

$$\theta^* = \frac{T - T_\infty}{T_i - T_\infty} = \sum_{n=1}^{\infty} C_n \exp(\vartheta_n^2 Fo) J_0(\vartheta_n r^*) \quad (1)$$

This problem is also numerically simulated in ANSYS FLUENT and results are compared to the analytical solution presented in equation 1 above. The soda can was simulated as a long cylinder (Diameter = 6.6 cm). The end effects were neglected. Figure 1 below shows the geometry of the can together with the adopted boundary conditions.

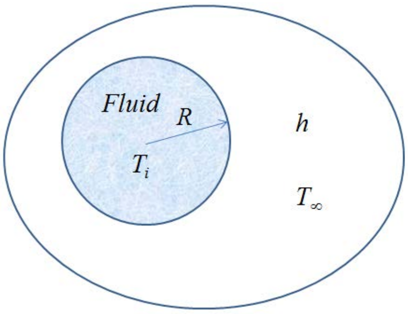


**Figure 1:** A schematic of the geometry and boundary conditions

Figure 2 below shows the geometry of the studied problem as simulated in ANSYS FLUENT.

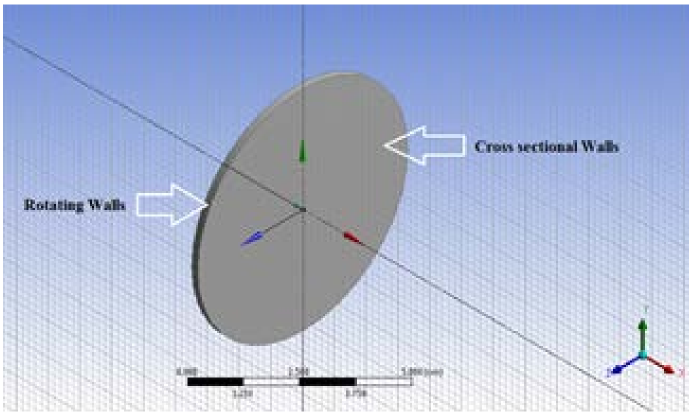


**Figure 2:** 3D geometry of the full disk as simulated in ANSYS FLUENT

### *2.2. Numerical Study*

A circular computational domain was used in this study to perform the numerical calculations. The cylindrical coordinate system was adopted with the origin coinciding with the center of the rotating container. The flow is assumed to be two-dimensional, incompressible, transient and laminar. The working fluid is soda with constant physical properties as follows:

- Density $\rho = 1000\ kg/m^3$
- Dynamic viscosity $\mu = 0.001731\ kg/m.s$

- Specific heat $C_p = 4178 \ J/kg.K$
- Thermal conductivity $k = 0.55 \ W/m.K$

The can was considered an infinite cylinder, so the temperature varies with respect to the radius r and the time t. The buoyancy effects were neglected.

The governing equations in this study are the continuity, momentum and energy equations and are expressed in cylindrical coordinates as follows:

$$\frac{\partial \rho}{\partial t} + \frac{1}{r}\frac{\partial}{\partial r}(r\rho V_r) + \frac{1}{r}\frac{\partial}{\partial \theta}(\rho V_\theta) + \frac{\partial}{\partial z}(\rho V_z) = 0 \tag{2}$$

$$\begin{aligned}&\frac{\partial V_\theta}{\partial t} + V_r\frac{\partial V_\theta}{\partial r} + \frac{V_\theta}{r}\frac{\partial V_\theta}{\partial \theta} + V_z\frac{\partial V_\theta}{\partial z} + \frac{V_r V_\theta}{r} \\ &= \frac{-1}{\rho r}\frac{\partial P}{\partial \theta} + \upsilon\left[\frac{1}{r}\frac{\partial}{\partial r}\left(r\frac{\partial V_\theta}{\partial r}\right) + \frac{1}{r^2}\frac{\partial^2 V_\theta}{\partial \theta^2} + \frac{\partial^2 V_\theta}{\partial z^2} - \frac{V_\theta}{r^2} + \frac{2}{r^2}\frac{\partial V_r}{\partial \theta}\right] + f_\theta\end{aligned} \tag{3}$$

$$\begin{aligned}&\frac{\partial V_r}{\partial t} + V_r\frac{\partial V_r}{\partial r} + \frac{V_\theta}{r}\frac{\partial V_r}{\partial \theta} + V_z\frac{\partial V_r}{\partial z} - \frac{{V_\theta}^2}{r} \\ &= \frac{-1}{\rho}\frac{\partial P}{\partial r} + \upsilon\left[\frac{1}{r}\frac{\partial}{\partial r}\left(r\frac{\partial V_r}{\partial r}\right) + \frac{1}{r^2}\frac{\partial^2 V_r}{\partial \theta^2} + \frac{\partial^2 V_r}{\partial z^2} - \frac{V_r}{r^2} - \frac{2}{r^2}\frac{\partial V_\theta}{\partial \theta}\right] + f_\theta\end{aligned} \tag{4}$$

$$\rho c_p\left(\frac{\partial T}{\partial t} + V_r\frac{\partial T}{\partial r} + \frac{V_\theta}{r}\frac{\partial T}{\partial \theta} + V_z\frac{\partial T}{\partial z}\right) = k\left[\frac{1}{r}\frac{\partial}{\partial r}\left(r\frac{\partial T}{\partial r}\right) + \frac{1}{r^2}\frac{\partial^2 T}{\partial \theta^2} + \frac{\partial^2 T}{\partial z^2}\right] + \Phi \tag{5}$$

Where the body forces are negligible, $f_\theta = 0$ .

The associated boundary and initial conditions used to solve Equations 2 through 5 are described as follows:

- At the rotation wall (at r = R):
  - For the thermal boundary condition, a uniform constant heat transfer coefficient was imposed:

$$-k\left.\frac{\partial T}{\partial r}\right|_{r=R} = h(T_s - T_\infty) \tag{6}$$

  - For the hydrodynamic boundary conditions, a no-slip boundary condition was imposed at the surface of the cylinder where the velocities at the surface (at r = R), were taken as follows:

$$V_\theta = V_r = 0 \tag{7}$$

- The initial conditions (at time t = 0) were used as follows:

$$T(r,\theta,0) = T_i \tag{8}$$

$$V_\theta = V_r = 0 \tag{9}$$

Numerical simulations were performed using the general purpose CFD code, ANSYS FLUENT, in order to solve Equations 2 through 5 together with the given boundary and initial conditions presented in Equations 6 through 9. A non-uniform grid system was used in all the simulations as shown in Figure 3 below. The domain was divided into 10348 elements, with a finer grid near the wall. A grid independence study was completed. As a result, the solution in this problem was independent of time-step and grid size. The convective terms were discretized with second-order upwind scheme. The convergence criteria were set to $10^{-4}$ for the momentum equations and $10^{-6}$ for the energy equations.

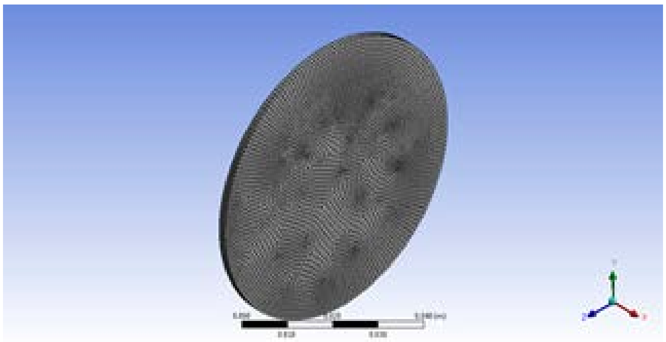

**Figure 3:** Non-uniform grid

### *2.3. Validation Problem Results and Discussions*

The investigated problem above was observed to mainly depend on the conduction resistance in the container since at lowest Biot number the convection resistance is very small. Therefore, an increase in the heat transfer convection coefficient barely affects the temperature distribution inside the container. Based on the analytical solution, it was found that it takes about twenty minutes to reach an average temperature of 11℃ if the initial temperature is 24℃.

The numerically calculated temperature profiles were compared to the analytical solution presented in Equation 1, and results are shown in Figure 4 below. An excellent agreement was obtained between the numerical and analytical solutions of this problem as shown in the figure below.

## 3. Heat Transfer from a Fluid in a Rotating Cylinder

This section of the study focuses on the flow and thermal solutions inside the rotating container. Initially, the velocity and temperature distributions of a fluid in a rotating cylinder are investigated under steady conditions.

An analytical solution for the velocity distribution inside the rotating cylinder was obtained. An attempt to obtain the temperature distribution for this problem was also conducted. A partial differential equation governing the temperature distribution together with the corresponding boundary conditions was obtained. An explicit analytical expression for the temperature distribution under these conditions was not obtained, however those temperature distributions were obtained numerically.

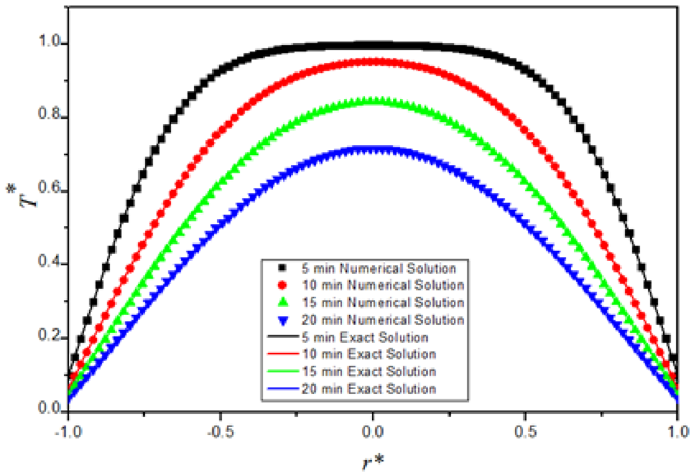


**Figure 4:** Comparison between the numerical and analytical temperature distributions at different time steps

In real life applications, the velocity and temperature distributions in a rotating container evolve from prescribed initial profiles to final steady state conditions over a non-negligible time period that depends on the fluid properties and the prescribed boundary conditions.

During this evolution time period, the velocity field develops from zero to a steady profile. However, the analytical solution for the temperature distribution for this problem during this evolution time period is a challenging task since the energy equation involves the velocity field which is also function of time and space. Therefore, the numerical approach was adopted to solve the temperature distribution for this problem.

Figure 4 below shows a sketch of the problem being investigated.

### *3.1. Anlytical Solution*

The velocity distribution in a fluid in a long cylinder rotating at constant velocity is solved analytically. The fluid is assumed to have constant properties with no axial motion or end effects. Since there is circular symmetry, the velocity distribution under steady state conditions becomes:

$$V_\theta = \left(\frac{r}{R}\right) V_w \qquad (10)$$

where $V_w$ is the velocity of the can wall, and *R* is the radius of the can.

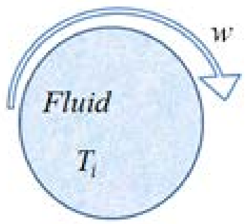


**Figure 4:** Geometry of the disk that is rotating at a uniform angular velocity

### *3.2. Numerical Study*

The same circular computation domain presented in section 2.2 was adopted in this problem. The fluid inside the cylinder is soda with the same fluid properties listed previously. Equations 2 through 5 were solved with the boundary and initial conditions described below.

- At the rotation wall (at r = R):
  - For the thermal boundary, this problem was initially solved with a uniform constant temperature:

$$T\big|_{r=R} = T_{wall} = 274.15K \tag{11}$$

  - The hydrodynamic boundary conditions were set as follows:

$$V_\theta\big|_{r=R} = wR = 0.33m/s \tag{12}$$

$$V_r\big|_{r=R} = 0m/s \tag{13}$$

The same problem was solved again with another thermal boundary condition at the wall. In this case, a convection boundary condition with constant heat transfer coefficient, *h*, was imposed at the rotating wall (at *r* = *R*) and is expressed as follows:

$$-k\frac{\partial T}{\partial r}\bigg|_{r=R} = h(T_s - T_\infty) \tag{14}$$

The initial boundary conditions (at *t* = *0*) were taken as follows:

$$T(r,\theta,0) = T_i = 298.15K \tag{15}$$

The numerical simulations were also conducted in ANSYS FLUENT. The same non-uniform grid described previously was used for this problem and the convergence criteria were set as described in section 2.2.

### *3.3. Results and Discussions*

The numerical velocity distribution is compared with the analytical expression obtained in Equation 10 and is shown in Figure 5 below. An excellent agreement is obtained between both analytical and numerical solutions of the angular velocity$V_\theta$. The numerical solution for the velocity distribution in the figure below was obtained after a long time in order to be able to assume that the flow has already reached its steady state velocity distribution.

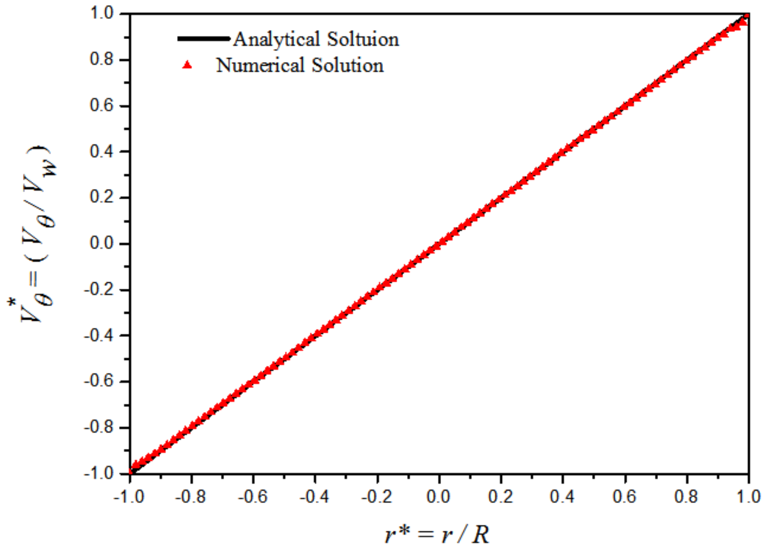


**Figure 5:** Steady state dimensionless velocity distribution

In real life applications, the can rotation causes a transient internal flow for a short time period. Figures 6 and 7 show the numerical solution of the velocity evolution at different dimensionless time steps. As shown in these plots, it takes the velocity field approximately $t^* = 20$ to reach steady state conditions.

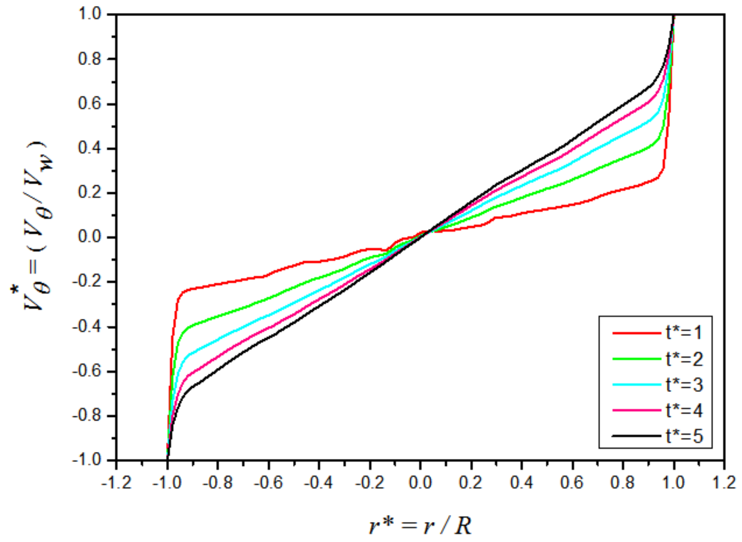


**Figure 6:** Dimensionless velocity distribution at different $t^*$

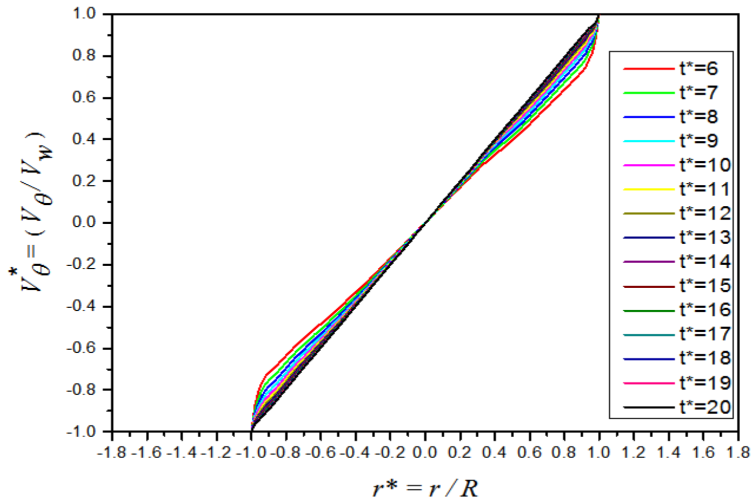


**Figure 7:** Dimensionless velocity distribution at different $t^*$

The partial differential energy equation was not solved explicitly for this problem due to its complexity. However a non-dimensional analysis was completed and yielded important results. Applying the hydrodynamic solution together with the adopted boundary conditions, the energy equation becomes:

$$\frac{1}{\alpha}\frac{\partial T}{\partial t} = \frac{1}{r}\frac{\partial}{\partial r}\left(r\frac{\partial T}{\partial r}\right) \tag{16}$$

Non-dimensionalizing the energy equation above using:

$$r^* = \frac{r}{R} \tag{17}$$

$$T^* = \frac{T - T_s}{T_i - T_s} \tag{18}$$

$$t^* = \frac{\alpha t}{R^2} = Fo \tag{19}$$

Results in the following non-dimensional energy equation:

$$\frac{\partial T^*}{\partial t^*} = \frac{1}{r^*}\frac{\partial}{\partial r^*}\left(r^*\frac{\partial T^*}{\partial r^*}\right) \tag{20}$$

The non-dimensional energy equation 20 is solved together with the following initial and boundary conditions for constant temperature:

$$T^*(r^*, 0) = 1 \tag{21}$$

$$\frac{\partial T^*}{\partial t^*}(0,t^*) = 0 \tag{22}$$

$$T^*(1,t) = 0 \tag{23}$$

And with the following initial and boundary conditions for constant heat transfer coefficient:

$$T^*(r^*,0) = 1 \tag{24}$$

$$-k\frac{\Delta T}{R}\frac{\partial T^*}{\partial r^*}\bigg|_{r^*=1} = h\Delta T\left(T_\infty^*\right) \tag{25}$$

$$-\frac{\partial T^*}{\partial r^*}\bigg|_{r^*=1} = \frac{hR}{k}\left(T_\infty^*\right) \tag{26}$$

Figures 8 through 11 show the temperature profiles at different times during the cooling process for the different adopted boundary conditions.

This problem with constant heat transfer coefficient was solved at two different Reynolds numbers which are 0 and 6291. The dimensionless profiles are compared in Figure 8, and the corresponding average decay with respect to time is compared in Figure 9.

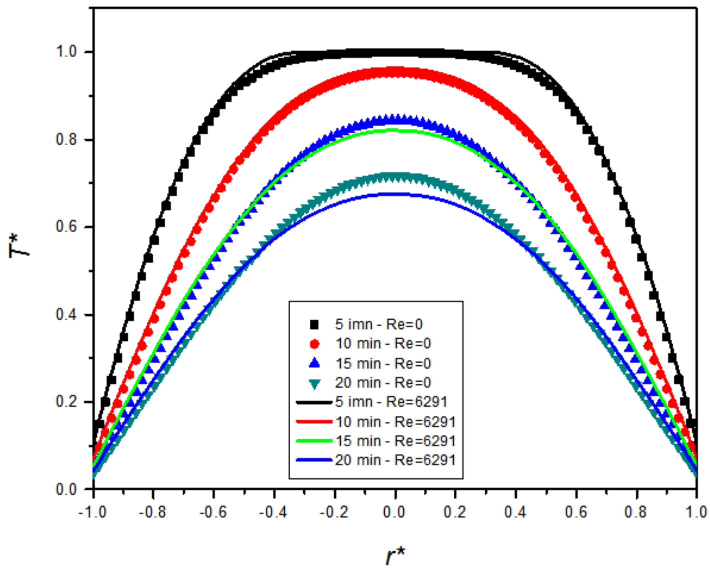


**Figure 8:** Dimensionless radial temperature distribution at constant heat transfer coefficient with 5 minutes interval time for two different Reynolds numbers (Re = 0; Re = 6291)

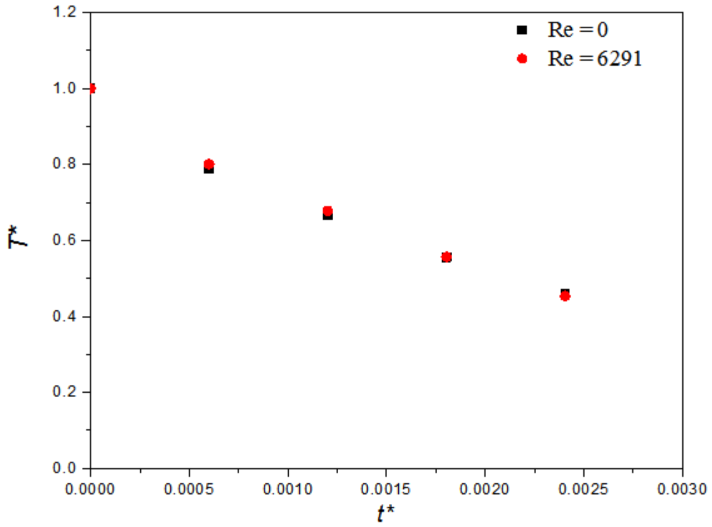


**Figure 9:** Average temperature distribution for different Reynolds numbers

The same illustrations are shown for constant temperature boundary conditions in Figures 10 and 11.

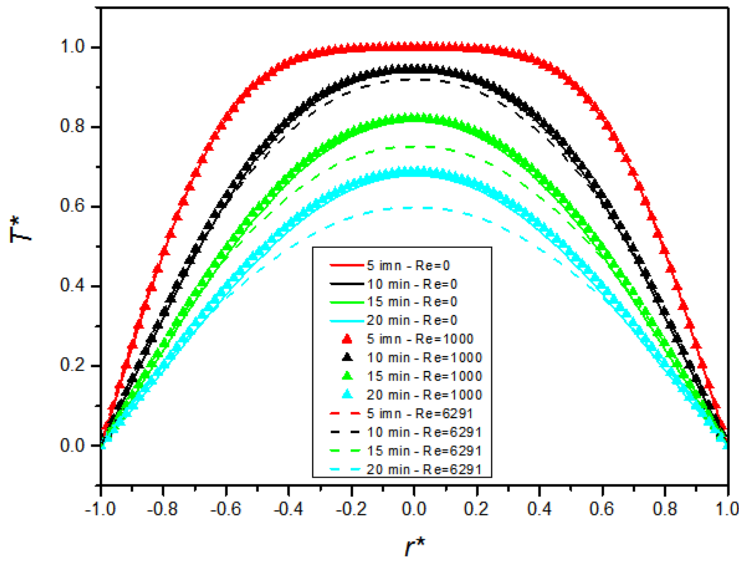


**Figure 10:** Comparison of the dimensionless radial temperature distribution at constant wall temperature with 5 minutes interval time for 3 different Reynolds numbers (Re = 0; Re = 1000; Re = 6291)

As shown in Figure 8, for constant heat transfer boundary conditions, rotating the container at a Re = 6291 showed a minor impact on the temperature distribution and no impact on the average temperature decay with respect to time as shown in Figure 9. For constant wall temperature boundary conditions, three Reynolds numbers were studied (0; 1000; 6291). No impact is seen on the temperature distribution at Re = 1000. However, when Reynolds number is increased to 6291, the average temperature decayed faster improving the cooling process as shown in Figure 11.

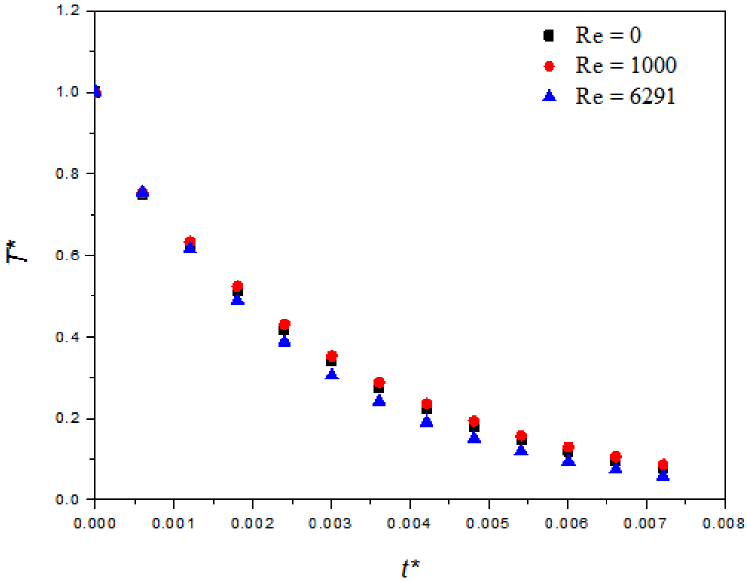


**Figure 11:** Average temperature distribution for different Reynolds numbers

## 4. Conclusions

The study presented in this paper focused on the fluid mechanics and heat transfer from a fluid in a rotating container. Analytical and numerical approaches were adopted. The analytical solutions were based on solving Navier-Stokes equations together with the specified boundary and initial conditions. ANSYS FLUENT was used to solve the problem numerically. A validation problem was initially solved and an excellent agreement was obtained between the numerical and analytical solutions. Heat transfer from a rotating cylinder in a fluid. This problem compared the analytical results for the flow and heat transfer to the numerical solutions conducted in ANSYS FLUENT. A perfect match between the two solutions of the velocity and temperature distributions was obtained for constant heat transfer coefficient and constant wall temperature boundary conditions. Heat transfer from a fluid in a rotating container was also solved for various thermal boundary conditions. This problem focused on the internal flow and thermal solutions inside the rotating container. The velocity distribution of a fluid that is rotating in an enclosure under steady state was solved analytically. The explicit equation for the temperature distribution under specific boundary conditions was not solved, but a partial differential equation was obtained. Numerical simulations were completed to solve the velocity and temperature distributions. For constant heat transfer coefficient and constant wall temperature boundary conditions, no major impact was found on the average temperature decay with respect to time while rotating the cylinder at different angular velocities.

## References


[1] A.G. Abdul Ghani, M.M Faird, S.J. Zarrouk, “The effect of can rotation on computational fluid dynamics.” Journal of Food Engineering, vol 57, pp. 9-16, 2003.

[2] F. Erdogdu, M. Tutar, “A computational study for axial rotation effects on heat transfer in rotating cans containing liquid water, semi-fluid food system and headspace.” International Journal of Heat and Mass Transfer, vol 55, pp. 3774-3788, 2012.

[3] A. Kannan, P.Ch. Gourisankar Sandak, “Heat transfer analysis of canned food sterilization in a still retort.” Journal of Food Engineering, vol 88, pp. 213-228, 2008.