FOSSEE, IIT Bombay
OpenFOAM Research Migration Project
April 21, 2022

Numerical Simulation of a food Sterilization Process involving Natural Convection Heating.

V Sai Yeshwanth
B.Tech student, IIT Bombay

Synopsis

In this research migration project, the Natural Convection heating-based sterilization process of canned liquid food was numerically studied, and the transient temperature and velocity profiles were generated. The temperature profiles were used to determine the time taken by the **Slowest Heating Zone (SHZ)** of the can to reach the sterilization temperature ($T_s = 100°C$), while the velocity profiles were used to track the motion of the SHZ inside the can.

The can is $0.107\,m$ tall and has a radius of $0.0419\,m$. It is heated from the cylindrical and bottom surfaces by condensing high pressure steam at a temperature, $T_c = 121°C$. Since the geometry (cylindrical can) and the external conditions (heat sources) are **axisymmetric**, the problem can be solved in two dimensions.

Water was used as the simulation fluid because it is the major constituent of many liquid foods. Its viscosity varies with temperature, and the variation was modelled using a polynomial equation ($\mu = aT + bT^2 + cT^3$). Similarly, the density variation was modelled using the **Boussinesq's approximation**. All other thermo-physical properties of water were assumed to be constant. For simulating the turbulence, the $k - \omega$ SST model was chosen over others owing to its reliability in both free stream and near wall regions.

The simulation was performed on **OpenFOAM v2012** and the data was post-processed on **ParaView 5.9.0**. The transient **buoyantBoussinesqPimpleFoam** solver was used for the simulation owing to its ability to accurately simulate buoyancy-driven turbulent natural convection flows. The results obtained by Ghani et. al.[1] using the commercial CFD code **PHOENICS** were used as a reference to assess the accuracy of the current simulation.

References

[1] AG Abdul Ghani, MM Farid, XD Chen, and P Richards. Numerical simulation of natural convection heating of canned food by computational fluid dynamics. *Journal of Food Engineering*, 41(1):55–64, 1999.

1 Introduction

Natural Convection Heating is employed in the food industry to heat liquid foods to their sterilization temperatures with the goal of extending their shelf life. The food is placed in cylindrical containers and heated from the bottom and cylindrical walls at 121°C by condensing high pressure steam . Since water makes up the majority constituent of many liquid foods, 'water' and 'liquid food' are used interchangeably in this report. The simulation is also performed using water as the fluid.

The heat supply is turned off when all the regions inside the container reach the sterilization temperature ($T_s = 100°C$). By knowing exactly when to stop the heat supply, one can prevent overcooking of food and reduce energy expenditures. But to accurately predict this heat supply stoppage time, a thorough understanding of the heating process is necessary. CFD simulations are generally performed to build that understanding.

One such simulation is performed by Ghani et. al.[1] where a commercial CFD code called PHOENICS is used. The transient temperature and velocity profiles of the heating process are generated and the heat supply stoppage time is calculated. In their simulation, the Boussinesq's Approximation is used to model fluid density variation with temperature while a second order polynomial equation is used to model the fluid viscosity variation with temperature. The turbulence in the fluid is modelled using the k-w SST turbulence model. The current research migration project aims to replicate the results obtained by Ghani et. al.[1] by performing similar simulations using OpenFOAM v2012.

2 Governing Equations and Models

The mathematical equations representing the mass, momentum, and energy conservation principles were solved using OpenFOAM v2012 to obtain the transient velocity and temperature profiles. Apart from these equations, the mathematical models describing the turbulence in the fluid and the fluid viscosity variation were also solved.

2.1 Governing Equations

The governing equations are a set of three partial differential equations representing the principles of mass, momentum and energy conservation. Since the current problem is axisymmetric, the governing equations have been written using the cylindrical coordinates. To account for the variation of fluid density with temperature, the Boussinesq's Approximation has been used. It is a commonly used assumption for buoyancy problems whereby the density variations are not explicitly modelled but their effect is represented by a buoyancy force which is proportional to the temperature variation.

Boussinesq's Approximation: $$\rho = \rho_0 - \alpha\rho_0(T - T_0) \tag{1}$$

Mass Conservation Equation: $$\frac{1}{r}\frac{\partial}{\partial r}(r\rho v) + \frac{\partial}{\partial z}(\rho u) = 0 \tag{2}$$

Energy Conservation Equation: $$\frac{\partial T}{\partial t} + v\frac{\partial T}{\partial r} + u\frac{\partial T}{\partial z} = \frac{k}{\rho C_p}\left[\frac{1}{r}\frac{\partial}{\partial r}\left(r\frac{\partial T}{\partial r}\right) + \frac{\partial^2 T}{\partial z^2}\right] \tag{3}$$

Momentum Conservation Equation in axial direction with Boussinesq's Approximation:
$$\rho\left(\frac{\partial u}{\partial t}+v\frac{\partial u}{\partial r}+u\frac{\partial u}{\partial z}\right)=-\frac{\partial p}{\partial z}+\mu\left[\frac{1}{r}\frac{\partial(r\frac{\partial u}{\partial r})}{\partial r}+\frac{\partial^2 u}{\partial^2 z}\right]-g\alpha(T-T_0) \quad (4)$$

Momentum Conservation Equation in radial direction:
$$\rho\left(\frac{\partial v}{\partial t}+v\frac{\partial v}{\partial r}+u\frac{\partial v}{\partial z}\right)=-\frac{\partial p}{\partial r}+\mu\left[\frac{\partial}{\partial r}\left(\frac{1}{r}\frac{\partial}{\partial r}(rv)\right)+\frac{\partial^2 v}{\partial z^2}\right] \quad (5)$$

2.2 Turbulence Model

The hybrid $k-\omega$ SST model is used to simulate the turbulence in the fluid. This turbulence model combines the near wall capabilities of $k-\omega$ model with the free stream capabilities of the $k-\epsilon$ model to produce relatively good results in the entire simulation region. In this turbulence model, the eddy viscosity is redefined using the function F_2 to account for the Bradshaw's assumption in adverse pressure gradient regions. The blending function F_1 is used to blend together the $k-\omega$ and $k-\epsilon$ turbulence models.

Redefined kinematic eddy viscosity :
$$\nu_T=\frac{a_1 k}{\max(a_1\omega, SF_2)} \quad (6)$$

Turbulence Kinetic Energy Transport Equation :
$$\frac{\partial k}{\partial t}+U_j\frac{\partial k}{\partial x_j}=P_k-\beta^* k\omega+\frac{\partial}{\partial x_j}\left[(\nu+\sigma_k\nu_T)\frac{\partial k}{\partial x_j}\right] \quad (7)$$

Specific Dissipation Rate Transport Equation :
$$\frac{\partial \omega}{\partial t}+U_j\frac{\partial \omega}{\partial x_j}=\alpha S^2-\beta\omega^2+\frac{\partial}{\partial x_j}\left[(\nu+\sigma_\omega\nu_T)\frac{\partial \omega}{\partial x_j}\right]+2(1-F_1)\sigma_{\omega 2}\frac{1}{\omega}\frac{\partial k}{\partial x_i}\frac{\partial \omega}{\partial x_i} \quad (8)$$

Closure Coefficients and Auxiliary Relations :

$$F_2=\tanh\left[\left[\max\left(\frac{2\sqrt{k}}{\beta^*\omega y},\frac{500\nu}{y^2\omega}\right)\right]^2\right] \quad (9)$$

$$P_k=\min\left(\tau_{ij}\frac{\partial U_i}{\partial x_j},10\beta^* k\omega\right) \quad (10)$$

$$F_1=\tanh\left\{\left\{\min\left[\max\left(\frac{\sqrt{k}}{\beta^*\omega y},\frac{500\nu}{y^2\omega}\right),\frac{4\sigma_{\omega 2}k}{CD_{k\omega}y^2}\right]\right\}^4\right\} \quad (11)$$

$$CD_{k\omega}=\max\left(2\rho\sigma_{\omega 2}\frac{1}{\omega}\frac{\partial k}{\partial x_i}\frac{\partial \omega}{\partial x_i},10^{-10}\right) \quad (12)$$

$$\phi=\phi_1 F_1+\phi_2(1-F_1) \quad (13)$$

$$\alpha_1=\frac{5}{9},\alpha_2=0.44$$

$$\beta_1=\frac{3}{40},\beta_2=0.0828$$

$$\beta^*=\frac{9}{100}$$

$$\sigma_{k1}=0.85,\sigma_{k2}=1$$

$$\sigma_{\omega 1}=0.5,\sigma_{\omega 2}=0.856$$

2.3 Transport/Viscosity Model

A transport model describes the rate at which a local momentum disturbance in the fluid is communicated to the entire fluid. It gives us a mathematical equation which relates the viscosity of the fluid with the local strain rate, temperature, etc. One such model is the power law model which is shown below in Equation 14. In low strain rate problems, the viscosity can be assumed to be independent of strain rate, i.e n=1. k is a constant called the consistency index whose dependency on temperature can be modelled using a second degree polynomial as shown in Equation 15. For this project, these two equations are combined to obtain the Equation 16 which is used to model the viscosity variation with temperature.

Power Law Viscosity Model: $$\mu = k\left(\frac{\partial u_i}{\partial x_i}\right)^{n-1} \quad \text{n=1 for low shear strain problems} \tag{14}$$

Dependency of constant k on temperature: $$k = a + bT + cT^2 \tag{15}$$

Combining the above two equations: $$\mu = a + bT + cT^2 \tag{16}$$

3 Simulation Procedure

3.1 Geometry and Mesh

Natural Convection Heat Transfer in a cylindrical can is a three dimensional problem. But the axisymmetry of the geometry (cylinder) and external conditions (heating from cylindrical surface) allows us to convert it into a two dimensional problem. Therefore, the entire physics of the problem can be captured on a $2D$ rectangular surface. The length of this surface will be equal to the height of the can, $h = 0.107\,m$, while the breadth of this surface will be equal to the radius of the can, $r = 0.0419\,m$ as shown in Figure 1.

To create this geometry and subsequently mesh it, the blockMesh utility available in OpenFOAM v2012 was used. The geometry was meshed using a total of 3519 nodal points with 69 in axial and 51 in radial directions. The mesh was deliberately made finer near the walls by using the Multi-grading option available in the blockMesh utility. In the axial direction, the thickness of the fine mesh was $13\,mm$ on both top and bottom walls with an expansion ratio of 2. Similarly in the radial direction, the thickness of the fine mesh near the wall was $10\,mm$ with an expansion ratio of 2. A schematic of the mesh is shown in the Figure 2.

The boundary layer thickness in the axial and radial directions was found out to be $5\,mm$ and $8\,mm$ respectively. This means that the fines mesh used in this simulation completely covers the boundary layer therefore adequately resolving it.

3.2 Initial and Boundary Conditions

It is crucial to specify the initial and boundary conditions of the problem we are solving. Failing to do so will produce a general solution which is applicable to all problems. In the current problem, the initial and

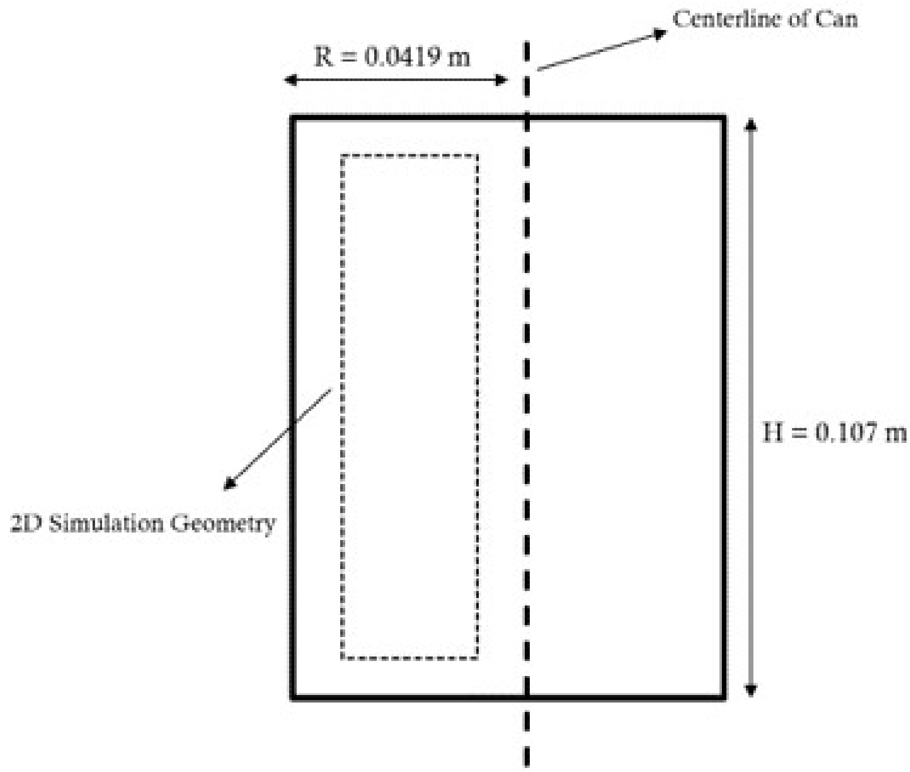


Figure 1: Schematic of the Geometry used for simulation

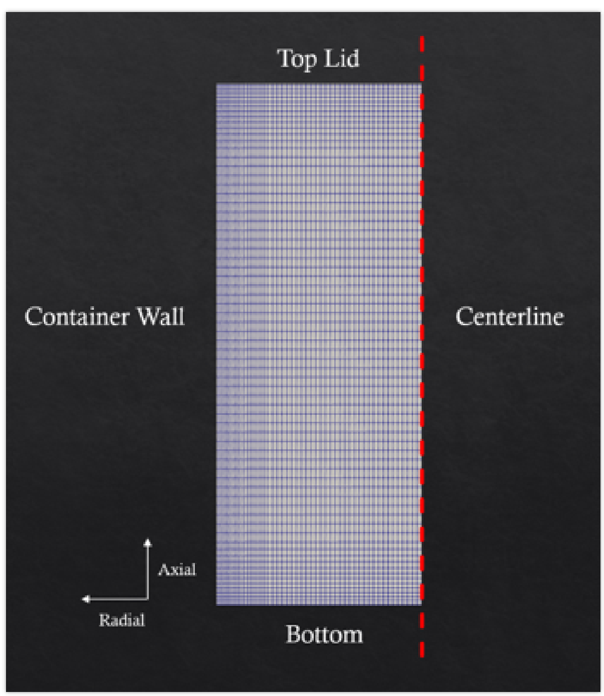


Figure 2: Schematic of the Meshing used for simulation

boundary conditions for temperature and velocity are specified.

Initial Conditions
The can is initially at room temperature ($T_r = 20°C$) while the fluid is at rest.

Boundary Conditions
For velocity, the no slip boundary condition is used on the top, bottom, and side walls while the axisymmetric boundary condition is used on the center-line.

For temperature, since steam is condensed at a constant temperature of $121°C$ at the bottom and side walls, a Dirchlet type boundary condition is used on them. The top wall of the can is insulated from the outside by a thin layer of air. Therefore a zero gradient boundary condition is applied on it. On the center-line, similar to the velocity case, an axisymmetric boundary condition is used.

3.3 Solver

The following physics would have to be simulated to get an accurate solution:

1. Buoyancy driven natural convection heating
2. Turbulence of the fluid
3. Fluid viscosity variation with temperature

The above aspects combined with the necessity to perform a transient simulation factored into the decision to choose the buoyantBoussinesqPimpleFoam solver.

4 Results and Discussions

4.1 Velocity Solution

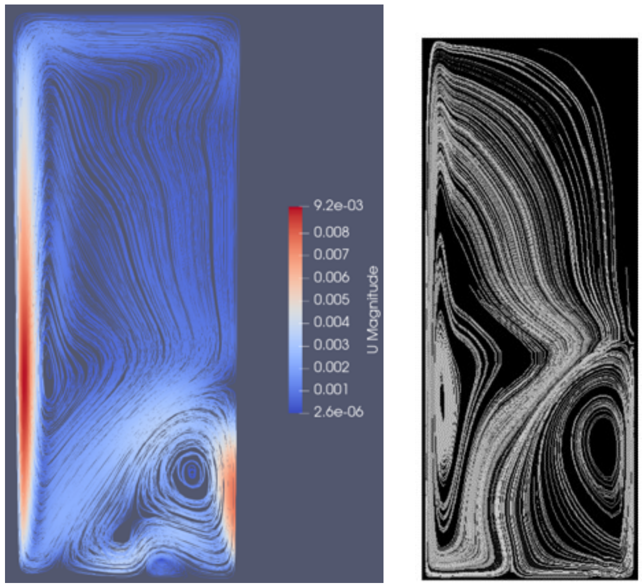
The velocity profile produced by this simulation after 180 seconds of heating is compared with that of Ghani et. al.[1] as showin the Figure 3a and 3b. The shapes of the velocity profiles are very similar. As seen in both figures, the heating from the side walls resulted in a reduction of density of the near wall fluid, causing it to rise upwards and hit the upper wall, after which it makes its way downward along the axis of the cylinder. In the velocity profile obtained by the research paper, a secondary vortex is also seen at the bottom of the can which is due to the heating from the bottom wall. This aspect of the velocity profile was also successfully reproduced in this simulation.

Validation of the velocity solution

To assess the accuracy of the solution obtained, the velocity and thickness of the upward rising fluid column was compared. While this simulation showed an $8\,mm$ thick fluid column rising at a speed of $31\,mm/s$ as shown in the Figure 4, the research paper observed a $7\,mm$ thick column rising at $40\,mm/s$. This amounts to a discrepancy of 14.3 % in fluid column thickness and 22.5 % in fluid column speed. On average, there is an 18.4 % error in the velocity solution. This could be due to inaccuracies in the parameters used for the $k-\omega$ SST turbulence model.

4.2 Temperature Solution

The temperature solution obtained by this simulation after $20\,s$, $60\,s$, $120\,s$, and $160\,s$ of heating are compared with those obtained by Ghani et. al.[1] as shown in the Figures 5a, 5b, 5c, and 5d respectively.



(a) Simulation (b) Research Paper

Figure 3: Comparison of the velocity profiles after 180 seconds of heating

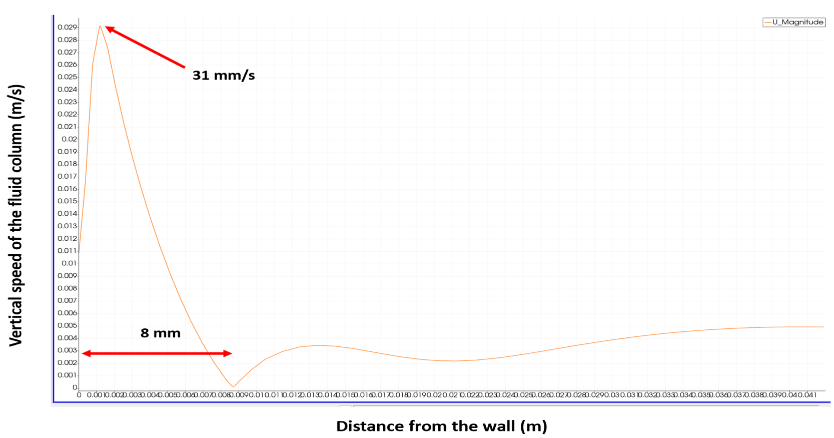
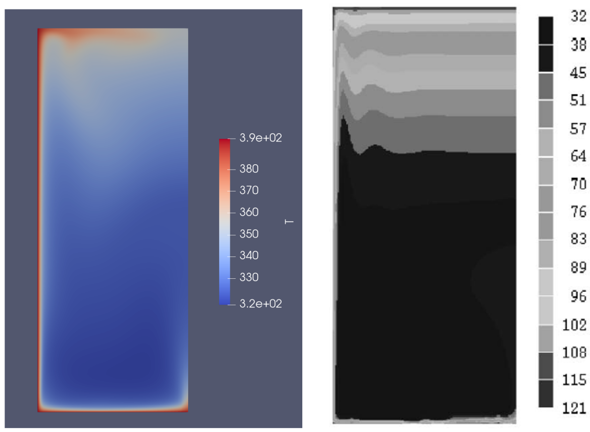
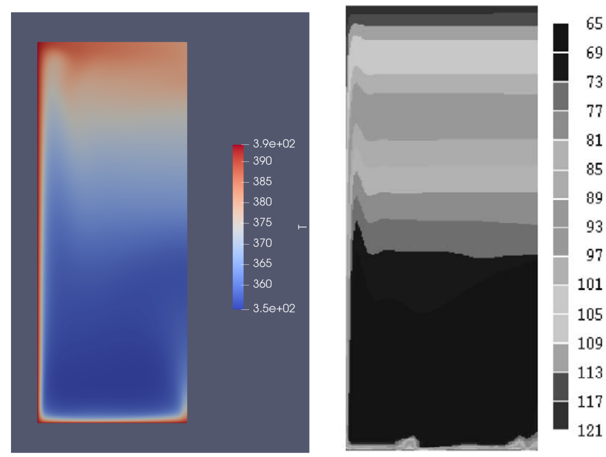


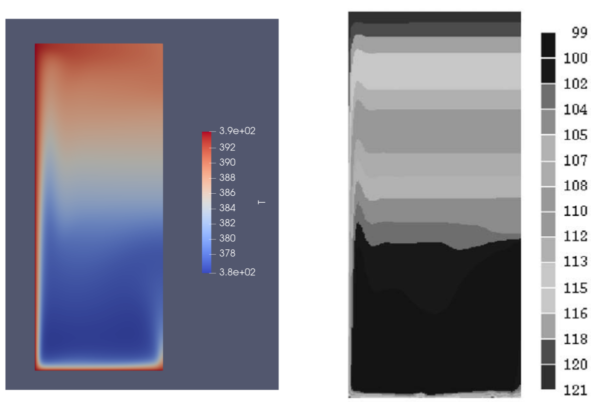
Figure 4: vertical speed of the fluid as a function of distance from the wall



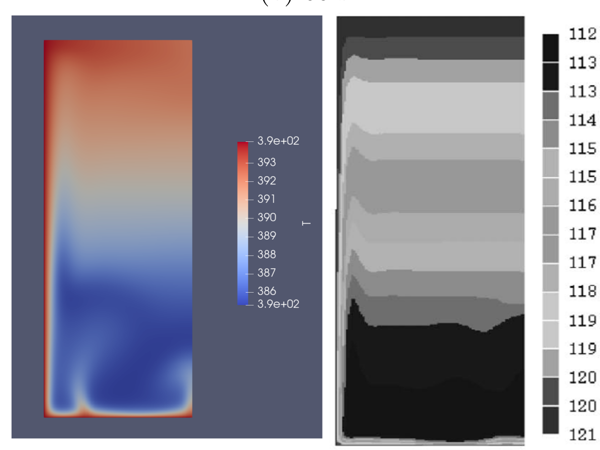
(a) 20 s



(b) 60 s



(c) 120 s



(d) 180 s

Figure 5: Comparison of Temperature Profiles at different instances of time (profiles obtained by the simulation are in colour while those of the research paper are in black and white)

The temperature profiles are very similar. The blue zone in each figure is the Slowest Heating Zone (SHZ) of the can which is seen moving downwards and shrinking as time progresses. This is expected because the rising hot fluid column near the side wall reaches the top wall, and then moves downwards from the top.

Validation of the temperature solution

The time taken by the fluid to reach the sterilization temperature is compared to gauge the validity of the temperature solution. While the simulation showed a sterilization time of 110 seconds, the research paper obtained a sterilization time of 120 seconds as shown in the Figure 6. This amounts to a discrepancy of 8.3 %. It should also be noted that this discrepancy in solutions is only seen in the transient states and gradually disappears as the system reaches its steady state.

References

[1] AG Abdul Ghani, MM Farid, XD Chen, and P Richards. Numerical simulation of natural convection heating of canned food by computational fluid dynamics. *Journal of Food Engineering*, 41(1):55–64, 1999.

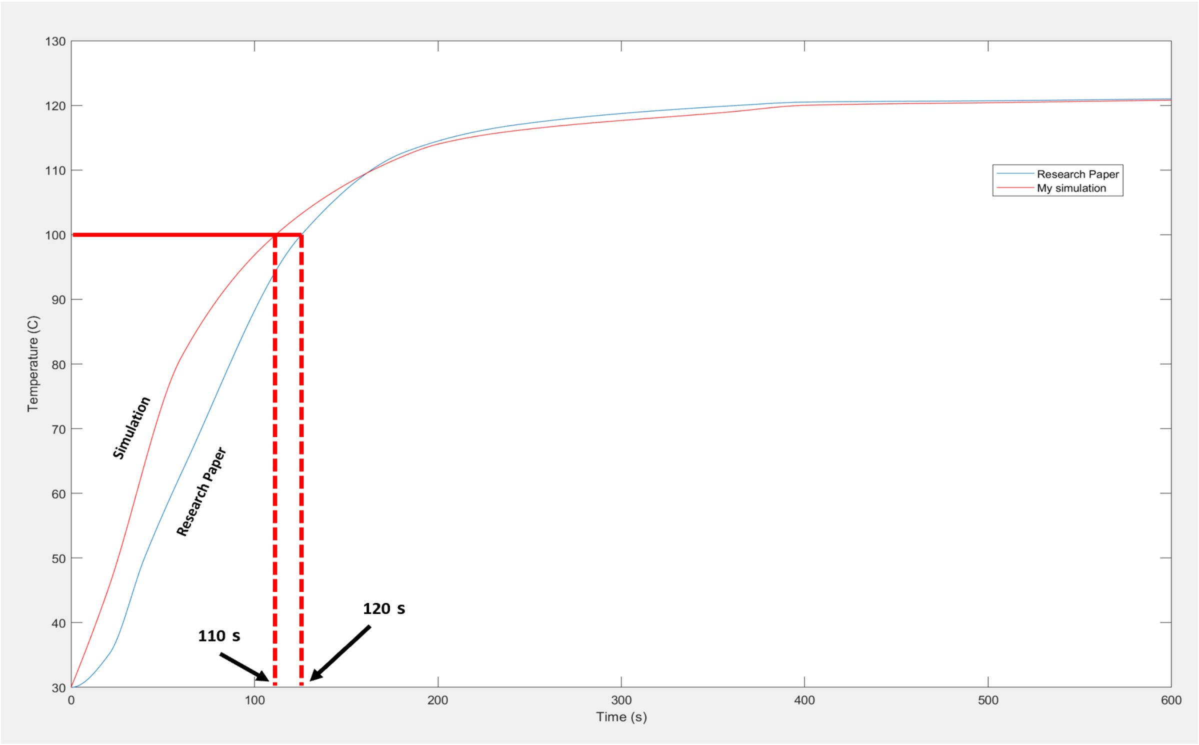


Figure 6: Comparison of Slowest Heating Zone temperatures as a function of time

DISCLAIMER: This project reproduces the results from an existing work, which has been acknowledged in the report. Any query related to the original work should not be directed to the contributor of this project.