A study of Rate of Temperature Change during Tire Curing Using Computational Fluid Dynamics

Watthanai Thamsamerdee¹, Kunthakorn Khaothong² and Jetsadaporn Priyadumkol¹,

¹Department of Mechanical Engineering, Faculty of Engineering, Mahidol University, Nakhon Pathom, 73170, Thailand
²Department of Mechanical Engineering, Faculty of Engineering at Kamphaeng Saen, Kasetsart University, Nakhon Pathom, 73140, Thailand

* Corresponding Author: jetsadaporn.pri@mahidol.ac.th

Abstract
The most important procedure in tire manufacturing is curing. The purpose of the curing is to apply high pressure and temperature over a period of time to unvulcanised rubber so that the rubber is cured to the point of best material properties. In this research, fluid flow analysis was considered to understand airflow pattern in an autoclave for a better quality of the product. We present preliminary results of the temperature distribution of heat transfer in an autoclave using Computational Fluid Dynamics package, OpenFOAM. The turbulent model was verified by comparison with the experimental results to obtain an accurate heat transfer coefficient. It was found that the error was 12%. The results from the simulation and experiments are in good agreement. Consequently, this turbulent model can be used to determine the heat transfer coefficient to achieve a better and more efficient curing process.

Keywords: Computational Fluid Dynamics, Curing, Heat transfer coefficient, Simulation

1. Introduction
The curing method is the longest in tire manufacturing. The tires are cured for a certain condition in order to get the properties. The curing process is accomplished under pressure and temperature provided by an autoclave. Autoclaves are cylindrical pressure chambers with doors to manufacture the rubber parts. The curing method is energy-consuming to minimize the cycle of time and understanding the behaviour of fluid flow in the curing method, the simulations were performed using CFD software.

Finite volume and finite difference methods have been widely used to solve CFD problems [1]. These methods require mesh generation in the simulation. It is very complicated for simulations, especially when the mesh has a complex and large shape. Flow simulation based on time and mesh during the simulation [2-4]. Computational fluid dynamics has been used by many researchers to analyse the problems of heat transfers and fluid flow. In 2013, Hamlin [5] presented the distribution of steam in an autoclave using CFD analysis through the ANSYS program. The SST k-ω model was used as turbulence...
modelling. The SIMPLE scheme was used to solve the pressure-velocity coupling calculations. The results showed that the model can be used to predict the general trends of steam distribution in the chamber. Ismail et al. [6] studied an optimization of the sterilization process. The experiment carried out temperature and time for two different types of retorts or autoclaves which were water immersion and water spray. The temperature distribution was considered to optimize performance. In 2015, Mosna and Vignail [7] used a k-ω SST turbulence model to predict the fluid flow and energy equation within the baking autoclave for sterilizing the packaging. The results showed that non-tray was heater than tray type, which was caused by the tray blocking the air flow. Chen et al. [8] used ANSYS to simulate temperature distributions of the frame mould for the autoclave forming process. The k-ε turbulence model was selected. The result showed that the maximum temperature appears at the corner of mould and the lowest is at the centre of the back. Their simulation reported accuracy of 9% for the temperature distribution. In 2017, Yongqian et al. [9] studied autoclave variables including pressure, velocity and heating rate. They proposed a 3D symmetries model with Boussinesq equation which use a constant density fluid model. It was found that increasing the velocity improves the temperature uniformity, while raising the pressure and the heating rate have a negative impact on the temperature uniformity. Gao et al. [10] compared the RNG k-ε model and standard k-ε model on simulating the unsteady flow. The results showed that the RNG k-ε was the high precision model compared with benchmark data. Bohne et al. [11] presented airflow and heat transfer inside the autoclave. The heat transfer was measured by lumped mass calorimeters. The effect of calorimeter measurements had been compared with the simulation results. The experiment and simulation results were 74 Wm⁻²K⁻¹ and 60.5 Wm⁻²K⁻¹, respectively. The simulation results showed a complicated turbulent flow pattern in the autoclave using the commercial software package ABAQUS CFD. Zhang et al. [12] compared various turbulent models for prediction of the airflow pattern and turbulent using FLUENT (version 6.2). These models were conducted to cover a wide range of CFD approaches which are Reynolds averaged Navier-Stokes (RANS), hybrid RANS, and detached eddy simulation (DES) and large eddy simulation (LES). This study used the relative error between numerical results and experimental results at measured significant criterion. The results show that if the error is less than 10%, the model accuracy is rated as A. Rating B is given to predictions with relative error of less than 20-30%. While rating C is given to the error about 30-50%. And if the error is larger than 50%, the model accuracy is rated as D.

In this study, the CFD was used to simulate using OpenFOAM to study the airflow pattern during curing processing. The heat transfer coefficient (HTC) results were validated against the value obtained from experiments [11]. To understand the behaviours of fluid flow, this method was also used to inspect the distribution of temperature inside the autoclave to develop the performance and decreasing the manufacturing times.

2. HTC measurement
The gas flow patterns in an autoclave are quite complex, for both empty and loaded autoclaves. Arafath et al. [13] proposed experiments to measure the heat transfer coefficient (or HTC) distribution at calorimeter inside the autoclave. The results showed that calorimeter was promising options to measure HTC values. The heat transfer coefficient (HTC) in an autoclave can be measured by [11]:

\[
h = \frac{\dot{m}c_p \Delta T}{\frac{T}{T_0} - 1} \frac{A}{\Delta T}
\]
where \( m \) is the mass of the aluminum plate, \( T \) refers to the autoclave fluid temperature, \( T_e \) the temperature within the calorimeter and \( A \) refers to the lower and upper free surfaces of the aluminum plate. The specific heat capacity \( C_p \) of the aluminum plate was determined using DSC-measurements and was modeled temperature-dependent

\[
c_p = (-0.0101T^2 + 7.8934T - 657.8658)
\]  

(2)

3. **Numerical model**

Temperature measurement was carried out in an autoclave, based on the experiment\[11\], to be used for validation with the computational result. The dimensions of the autoclave using SolidWorks 2010 program are shown in Figure 1.

![Figure 1: Dimensions and calorimeter position inside autoclave, in mm.](image1)

Computational fluid dynamics model of the autoclave was generated using OpenFOAM software. The transient analysis of the two-dimensional model was solved by the k-\( \omega \) SST turbulence model for the analysis of turbulent flow. The domain is a quadrilateral of 50,600 meshes. The non-dimensional near-wall grid distance \( (y^+) \) of the surface of the calorimeter is between 3.5-6 \[11\]. The lower the value, the more effective the skin is. The boundary condition is set to have 2 ways velocity inlet 5.058 m/s with temperature 333.15 K and outlet condition is set to be pressure outlet 700,000 Pa as shown in Table 1.

![Figure 2: Two-dimensional computational domain.](image2)
Table 1. Boundary conditions in an autoclave

| Boundary       | Setting                                           |
|----------------|---------------------------------------------------|
| Inlet 1 and 2  | Velocity inlet 5.058 m/s temperature 333.15 K    |
|                | Kinetic turbulent viscosity 2.7e-5 m²/s           |
| Outlet         | Pressure outlet 700,000 Pa                        |
| Calorimeter    | Temperature 323.15 K                             |
| Other          | Wall                                              |

4. Results and discussion

The results of the CFD simulation at various times in autoclave were reported. The post-processed result was compared with those obtained from the experiment at the height of 5 cm below the calorimeter for validation. The heat transfer coefficient (HTC) difference between those simulated and measured in this autoclave was obtained. The HTC values of the numerical and experimental were 74 Wm⁻²K⁻¹ and 82.9 Wm⁻²K⁻¹, respectively. It was found that the data error between the simulation and experimental data was 12%. Therefore, using available CFD software OpenFOAM can predict a phenomenon of turbulence flows in autoclave correctly.

Figure 3 shows the temperature contour at various times which are 20, 40, 60, 80, 100 and 106 seconds, respectively. In 20 seconds, the airflow inlets entered to the autoclave and clashed, causing the turbulence of the air (fig. 3a). As can be seen in Figure 3b-3e, the air began to split into two ways: the bottom and the top, after colliding with the object. The temperature of the bottom was lower than the top. The air above had a higher temperature below, but while the bottom was getting higher. Finally, the temperature reached the steady state condition at 106 seconds, as shown in fig. 3f.
5. Conclusion

We present preliminary results of a major systematic study of heat transfer in autoclaves using simple lumped mass calorimeters. A 2-D numerical simulation has been conducted to study the flow pattern. The result of CFD analysis with k-ω SST model had converged in terms of HTC. The k-ω SST model is an enhancement of the original k-omega model. It has the advantage that applied the viscous-affected region without further adjustment [14]. The result has been compared to experiment and appear to be in a good agreement with the experimental data [11]. It has been confirmed that the proposed model can be used for prediction of temperature distribution. The results show that the error was 12%, which suggests that the simulation using the OpenFOAM program of this research is more accurate than the simulation [11]. In the future, a 3-D calculation should be made and the results will be compared to experiment. Then, it will be possible to minimize the cycle time in the curing process.

Acknowledgments

The authors are grateful to the Laboratory of Computer Mechanics for Design (LCMD), Department of Mechanical Engineering, Mahidol University for valuable technical support.

References

[1] Pletcher R Tannehill J Anderson D 2013 Computational Fluid Mechanics and Heat Transfer (CRC Press)
[2] Smirnov P Zaitsev D 2004 St. Petersburg Polytec. Un. J. Eng Sc. Techn. 36 70–81
[3] Zaitsev D Schur N 2006 St. Petersburg Polytec Un. J. Eng Sc. Techn. 47 15–22
[4] Zaitsev D Smirnov P Yakubov S Balashov M 2012 Software and Systems 2 32–35
[5] Hamlin P 2012 Master of Science Thesis, Chalmer’s University of Technology
[6] Ismail I Fahmy A Azab A Abadir M Fateen S-E 2013 J. Ag and Vet Sc. 6(4) 26–33
[7] Mosna D Vignali G 2015 J. Food Eng. 11(6) 715–729
[8] Chen F Zhan L Xu Y 2015 Iran Polymer J. 24 927–934
[9] Yongqian X Lihua Z, Minghui H Rulin S 2017 Procedia Eng. 207 269–274
[10] Gao F Wang H, Wang H 2017 Procedia Eng. 205 3970–3977
[11] Bohne T Frerich T Jendrny J Jürgens J P Plosnikhin V 2018 J. Composite Materials 52(12) 1677–1687
[12] Zhao Z Wei Z Zhiqiang Z Chen Q 2007 HVAC&R Research 13 14–15
[13] Slesinger N Shimizu T Arafath A R A Poursartip A 2009 Proc. the 17th International Conference of Composite Materials 1–10
[14] El- Behery S Hamed M 2009 Engineering and Technology 53 10–11