Energy efficient cooking pot- Ecopot

add all team members

May 2021

1 Introduction

Ecopot is an attempt to make a more fuel-efficient cooking pot for the one billion people who cook on wood gathered by hand from the ground. Making a pot which requires less firewood would decrease the world carbon footprint, but more importantly would save many people, often women, hours of tedious labor and may help reduce gender inequity.

ADD A LITTLE MORE INFO

2 Choosing the right fin shape

Convective heat transfer between a surface and the surrounding has been a major issue and a topic of study for a long time. In this project, the heat transfer performance of fin is analyzed using ANSYS workbench for the design of fin with various design configuration such as cylindrical configuration, square configuration and rectangular configuration. The heat transfer performance of fin with same base temperature having various geometries is compared. In this thermal analysis, Aluminum was used as the base metal for the fin material and for various configurations. Fin of various configuration are designed with the help of Solidworks and analysis of fin performance is done through ANSYS Fluent.

The Fin is a major component used in many systems for increasing the rate of heat transfer. By doing thermal analysis on the fins, it is helpful to know the heat dissipation and rate of heat transfer in different types of fins. Increasing the temperature difference between the fin configuration, slightly increasing the convection heat transfer coefficient or slightly increasing the surface area of the fin increases the heat transfer. Sometimes it is not economical or it is not feasible to change the first two options. Therefore we try to compare the performance of fins by changing the surface area i.e the shape of the fins.

2.1 Simulation parameters

1. Rectangular fin: Volume = $4X10^{-5}m^3$, Area = $4X10^{-4}m^2$, Mesh - Fine mesh, Model - Energy (ON), Viscous k-epsilon (Realizable), Boundary conditions - Inlet velocity of 5 m/s, heat transfer coefficient = 210 W/m-K (Aluminum), Solution method - SIMPLE algorithm, Initialisation - Standard, Number of iterations - 500

2. Circular fin: Volume = $4.15X10^{-5}m^3$, Area = $4.15X10^{-4}m^2$, Mesh - Fine mesh, Model - Energy (ON), Viscous k-epsilon (Realizable), Boundary conditions - Inlet velocity of 5 m/s, heat transfer coefficient = 210 W/m-K (Aluminum), Solution method - SIMPLE algorithm, Initialisation - Standard, Number of iterations - 500

3. Triangular fin: Volume = $2X10^{-5}m^3$, Area = $2X10^{-4}m^2$, Mesh - Fine mesh, Model - Energy (ON), Viscous k-epsilon (Realizable), Boundary conditions - Inlet velocity of 5 m/s, heat transfer coefficient = 210 W/m-K (Aluminum), Solution method - SIMPLE algorithm, Initialisation - Standard, Number of iterations - 500

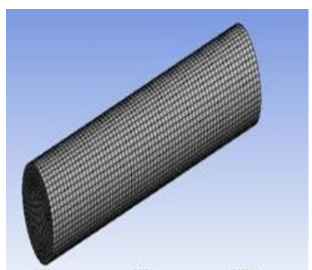

Figure 1: Meshed circular fine

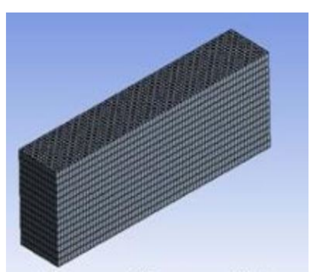

Figure 2: Meshed rectangular fine

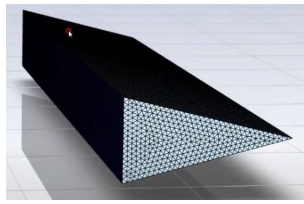

Figure 3: Meshed triangular fine

Simulations were performed for 500 iterations and convergence was observed around 400 iterations.

2.2 Simulation results

The results of the simulation along with the colorscale is shown below. The base temperature of the fin was initialised to 380 K (100 °C). The following assumptions were used for the simulation:

- Steady state
- Constant material properties (independent of temperature)
- No internal heat generation
- One-dimensional conduction
- Uniform cross-sectional area
- Uniform convection across the surface area

Figure 4: Heat transfer colorscale

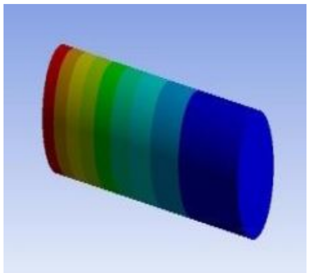

Figure 5: Heat transfer across circular fin

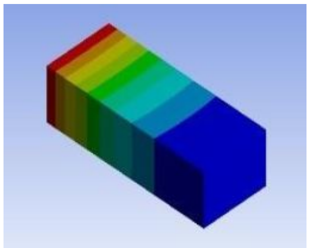

Figure 6: Heat transfer across rectangular fin

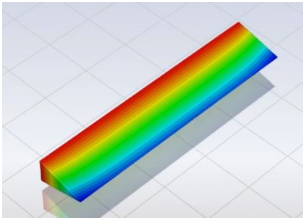

Figure 7: Heat transfer across triangular fin

2.3 Conclusion

The use of fins (extended surface), provide efficient heat transfer. Heat transfer through fin of triangular configuration is higher than that of other fin configurations. Temperature at the end of fin with triangular configuration is minimum, as compare to fins with other types of configurations. The effectiveness (defined as the ratio of the actual heat transfer that takes place from the fin to the heat that would be dissipated from the same surface area without fin) of fin with triangular configuration is also greater than other configurations. Choosing the optimum sized fin of triangular configuration will reduce the cost for heat transfer process and also increase the rate of heat transfer.

3 Designing the top of the pot

As per multiple field surveys, people prefer pots having a rounded top to latch onto rather than having separate handles attached to the top. This requires changing the design of the pot at the top and studying the heat transfer around that region.

3.1 Simulation parameters

Mesh - Fine mesh, Model - Energy (ON), Viscous k-epsilon (Realizable) with Enhanced wall treatment, Boundary conditions- Pressure based, Absolute, Time-Steady, 2D Planar, Wall - No slip, Solution method - SIMPLE algorithm, Initialisation - Standard, Number of iterations - 500

3.2 Simulation results

3.2.1 Simple extended top

Figure 8: Temperature contour at the top of the pot, iteration number 100

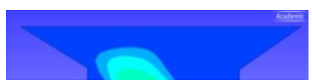

Figure 9: Temperature contour at the top of the pot, iteration number 300

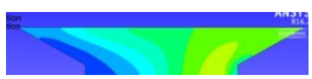

Figure 10: Temperature contour at the top of the pot, iteration number 500

3.2.2 Extended top with a rounded edge

Figure 11: Temperature contour at the top of the pot, iteration number 100

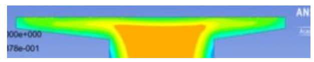

Figure 12: Temperature contour at the top of the pot, iteration number 300

Figure 13: Temperature contour at the top of the pot, iteration number 500

The colorscale is same as the scale used in section 2.2.

3.3 Conclusion

By comparing the temperature contours, we find that the top with rounded edges has a more uniform heat transfer rate across the profile and hence would lead to faster heating time. Also, since more heat is lost along the sides of the top, covering it using a suitable cover would improve the efficiency further.

However, making a rounded edge would increase the production time as it would be a two step process - making a simple extended top and then flattening the sides. However, this is a trade off that's worth making since it would lead to a more efficient pot.

4 Effect of fin configuration on turbulent natural convective heat transfer

We wish to study the difference in heat transferred due to different arrangement of fins radially. Effect of fin height on flow and temperature fields has been studied for different fin arrangements. Rahnama and Farhadi [5] reported this effect on turbulent natural convection for the case of horizontal fin arrangements. The streamline pattern obtained showed that increasing fin height to radius difference ratio more than 0.4 results in two recirculation zones.

The arrangements tested includes two fins in horizontal and two fins in vertical direction and also the same fins rotate 45 degrees relative to horizontal and vertical arrangement. Flow and temperature fields in the form of streamline and isotherm are shown superimposed. There are differences between the streamline and temperature contours for different configurations.

Buoyancy-induced motion in the lower part of the annulus is very weak due to the fact that high temperature surface (inner cylinder surface) is over the low temperature one (outer cylinder surface). So there is a stable density stratification which resists to fluid flow. In fact, there is a very weak fluid motion in the lower part of the annulus.

The fluid flow being started to circulate in the upper part of annulus, may have enough momentum to continue its motion to the lower part. The fluid along the inner cylinder tends to move away from the symmetry line and tries to bend downward upon meeting the first fin. However, the buoyancy force created by heating of air prevents air from moving downward and thus creates a region of almost stagnant flow in the bottom of the annulus.It should be mentioned that increasing the number of fins makes the number of recirculation regions to increase.

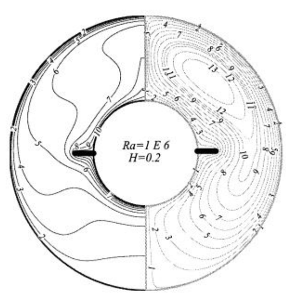

Figure 14: Isotherms for 2 fin arrangement

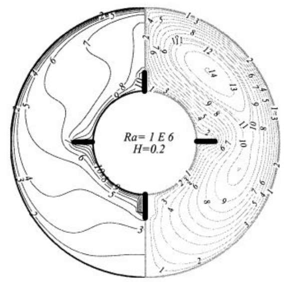

Figure 15: Isotherms for 4 fin arrangement, $\theta = 90°$

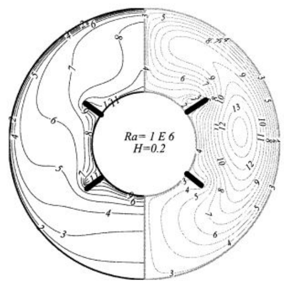

Figure 16: Isotherms for 4 fin arrangement, $\theta = 45°$

4.1 Nusselt Number

The Nusselt number (Nu) is the ratio of convective to conductive heat transfer at a boundary in a fluid. Convection includes both advection (fluid motion) and diffusion (conduction). The conductive component is measured under the same conditions as the convective but for a hypothetically motionless fluid.

Nu $= R_i ln(R).\frac{\partial T}{\partial r}$

where the last term represents the temperature gradient in radial direction.The mean Nusselt number is obtained by integrating the local Nusselt number over the inner cylinder and fin surfaces.

Figure shows variation of mean Nusselt number ratio for the case of two fin arrangements with ratio of fin height to radius difference.

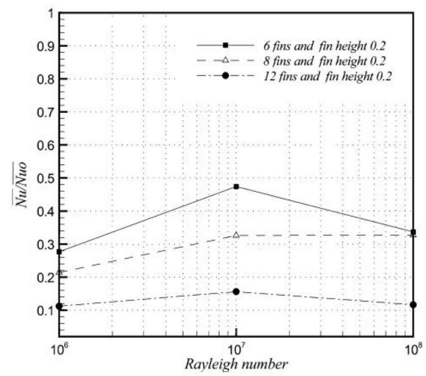

Figure 17: Variation of Nusselt number for different fin arrangements

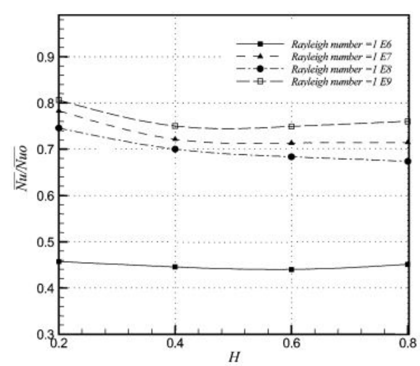

Figure 18: Variation of Nusselt number with Rayleigh number

4.2 Conclusion

The following conclusions can be drawn from the fin arrangement study:

1) For all configurations, results indicate that local Nusselt number increases with an increase in Rayleigh number. Increasing the number of fins increases mean Nusselt number ratio, with in turn increases heat transfer rate.

3) Higher fin height has some effect on fluid motion. If there is a tendency toward reducing heat transfer rate between two concentric horizontal cylinders, it is better to use higher fin height.

4) <u>Fin arrangement has no significant effect</u> on mean Nusselt number prediction although its effect on flow and temperature fields is remarkable for the case of four fin arrangement.

5) There is a possibility that alternating shorter and longer fins (of different heights) may increase the heat transfer rate due to an increase in the number of recirculation zones.

5 Studying the flow of air around the pot

We try a CFD approach for modelling of combustion in a cooking stove and study the flow of air around the stove. The stove has a cylindrical structure made of aluminium and is covered with an aluminium plate to model airflow. The combustion phenomena inside the stove is modeled using the Eddy-breakup combustion model and the k- turbulence model.The flow considered, is due to the buoyancy effect and the spatial variation of species concentrations. Empirical data of a typical cook stove has been used as boundary conditions.

5.1 Boundary conditions

Conservation equations are solved by the numerical methods in which the boundary conditions will be the preparatory points of the equations. It is important to take the right boundary conditions that are most appropriate to the practical problem. There are three types of boundary conditions have been used for the simulations while getting a practical in- touch with the cook stove problem. The list of boundary conditions used are mentioned below.

Condition	Value
Wall influence on flow	No slip
Wall roughness	Smooth
Heat transfer	Domain interface
Emissivity	Grey color

Figure 19: Boundary conditions for airflow simulation

Computational domain and the physics of the simulation were built on the builder of the CFX-5.6. The conservation equations were discretized using the finite volume method in the solver. The SIMPLEC algorithm achieved velocity and the pressure relation. The total time duration for the simulation was 16 hrs 33 minutes.The results of the simulation are shown below.

To study the shape of airflow at the top, temperature and velocity data from the simulation region was dumped to an external file and was separately plotted using MATLAB. The velocity and temperature contours are shown below.

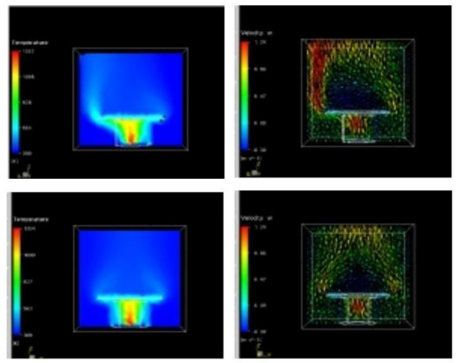

Figure 20: Temperature contours and velocity vectors of airflow around the pot

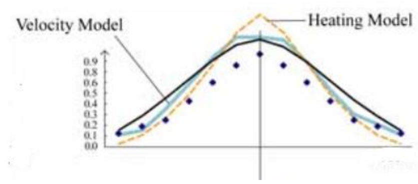

Figure 21: Plot of velocity vectors at the top of the pot

5.2 Conclusion

The following conclusions can be drawn from the airflow study:

1) Although the flow of air appears to take a hemispherical shape, the simulation eventually converges and a conical shape is seen above the pot.

2) The conical shape has been suspected to be formed due to the aerodynamics of the flow. When flow separates after travelling along the edges, the streams seems to undergo a separation and the two streams converge some region above the pot.

3) Fig 21 clearly shows the flow to be following an approach similar to flow around a moving aerodynamic vehicle. Hence if the shape of the pot lid is made similar to a cone, this would lead to air moving along the lid and result in maximum heat transfer.

6 CFD Analysis of a hemispherical cooking pot

We aim to analyse the double walled cooking unit under different geometry and flow conditions by using the CFD tool FLUENT 6.2. This study gives significant insights that will provide the direction for improvement in energy efficient cooking.

The model is created using Pro-E software and this model is meshed with a tetrahedral element, using hypermesh software. This meshed model is exported to the Fluent software for analysis.The following assumptions are made for the CFD analysis:
1) The density, specific heat and thermal conductivity are considered to be constant for the fluid in the pot
2) The outer surface of the cooking unit is perfectly insulated, and hence, the heat flux is considered as zero.

6.1 Boundary conditions

Velocity inlet (specified normal to the boundary, 0.5 to 2.5 m/s with increments of 0.5m/s), temperature (constant, applied to grid cells where back flow occurs), wall boundary condition with specified food side heat transfer coefficient and free stream temperature (no heat flux on the outside), food side heat transfer coefficient (50,60,2000 W/m^2-K). This large variation in heat transfer coefficient is considered due to various heat transfer mechanisms involved during any cooking process like free convection, stirring of the food stuff in the cooking unit, heat transfer due to nucleate boiling and evaporation, which has large variations in the heat transfer coefficient.

A separate Grid independence study was carried out for optimizing the mesh size and cell count. Fig. 13 shows the variation in the heat transfer rate with respect to the mesh count employed in the analysis. It was observed from the figure that after a mesh count of 520,000 tetrahedral elements, the variation in the heat transfer rate from the inner surface of the vessel was consistent. Hence, a standard mesh count of approximately 520,000 tetrahedral elements with a size of 2.4 mm was used for the final analysis, for all the vessel configurations. The Fluent software employs the finite volume method for solving the mass, momentum and energy conservation equations. The fully implicit method is adopted with the SIMPLE algorithm as the solver option for the steady flow simulation, and the second order upwind scheme was used for solving all the conservation and turbulence equations.

6.2 Simulation and results

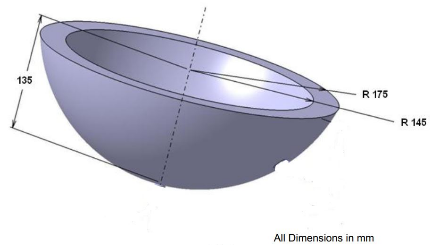

Figure 22: Model of a thin walled hemispherical pot

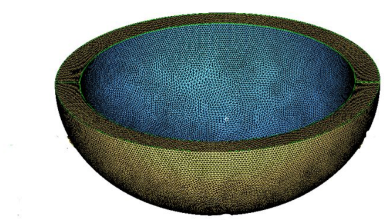

Figure 23: Meshed model of a thin walled hemispherical pot

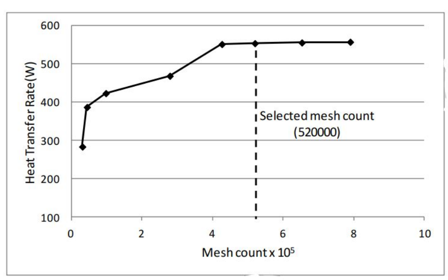

Figure 24: Finding the suitable mesh size

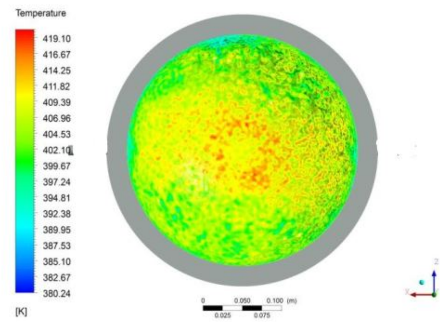

Figure 25: Temperature contour at 200 iterations

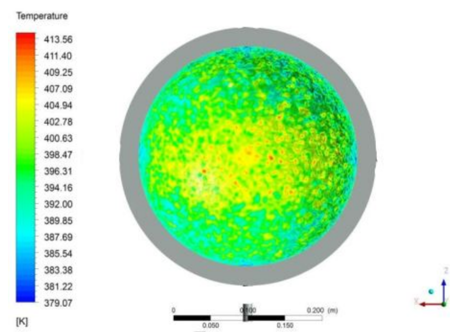

Figure 26: Temperature contour at 500 iterations

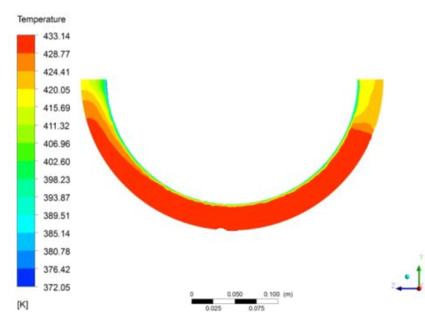

Figure 27: Temperature distribution within the wall

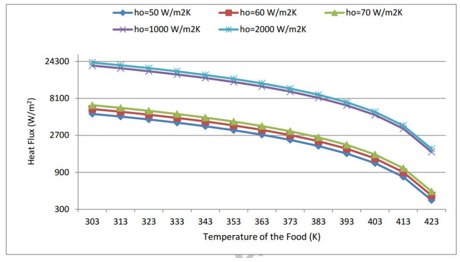

Figure 28: Temperature distribution of food within the pot

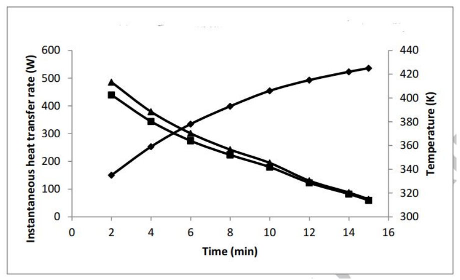

Figure 29: Heat transfer curve; triangle, square - heat transfer rate, different turbulent models, circle - temperature of water within the pot

From the simulation analysis performed at various surface heat transfer coefficient at the food side, it is inferred that the increase in the food side heat transfer coefficient (ho) plays a major role at the lower range of ho values, to increase the surface heat flux. However, at the higher range of ho values, the effect of the increase in the surface heat flux is insignificant.

7 References

[1] Knight RW, Goodling JS, Hall DJ. Optimal thermal design of forced convection heat sinks – analytical. J Electron Packag. 1991;113:313–321.

[2] Incropera FP, DeWitt DP. Fundamentals of Heat and Mass Transfer. New York: Wiley; 1990.

[3]Kundu B. A closed form solution for ac electro-kinetic driven flow in a microchannel. J Therm Eng. 2015;1:289–294.

[4]Mirapalli S, Kishore PS. Heat transfer analysis on a triangular fin. Int J Eng Trends Technol.. 2015;19:279–284.

[5] M. Rahnama, M. Farhadi, Effect of fin height on turbulent natural convection heat transfer in a horizontal annulus, in: The 13th International Symposium on Transport Phenomena, Victoria, BC, 2002, pp. 345–350.