CHAPTER 4

CFD MODELING AND ANALYSIS

Though different types of solar collectors, box type cooking units and other integrated direct type collector with cooking devices have been successfully developed in recent years, the associated units like the thermal storage system, cooking devices that use stored energy have not been given enough attention, and hence, have not been able to fully replace the conventional fuels. During the day time, there is no guarantee about consistent solar energy availability; therefore, to make use of the available solar energy for cooking applications efficiently, it is essential to design and develop a suitable cooking unit, that utilizes the heat energy from the heat transfer fluid (HTF), either directly from the solar collector or from the storage system. The current literature indicates that no such work has been attempted so far. The present work aims to fill this gap in the literature.

In the present work, an attempt has been made to develop an indirect type solar cooking unit in which the HTF from the storage tank is to be circulated through the annular portion of the unit at the time of cooking. There is no theoretically established procedure to assess the performance of such cooking units. Hence, in order to analyze the performance of the cooking unit, a computational fluid dynamics (CFD) simulation analysis has been carried out for both the tava and flat plate cooking units under similar geometrical and flow conditions used in the experimental investigation.
CFD is a powerful tool for fluid dynamics and thermal design in industrial applications, as well as in academic research activities. Selecting the relevant CFD package (such as CFX, Fluent, Flow 3-D and Phoenix) for a specific application, understanding the physics of the processes, introducing adequate simplifications and establishing an appropriate model are essential factors for obtaining reasonable results and correct thermal design. Optimizing the system experimentally, is expensive and time consuming, but the CFD is one of the robust methods for fluid flow simulation that reduces the time and cost significantly.

The present work is aimed at analyzing the tava and flat plate cooking unit under different geometric and flow conditions by using the CFD tool FLUENT 6.2. This study gives significant insights that will provide the direction for improvement in effective solar energy utilization.

This chapter reports the statement of the problem, modeling and meshing, and the computational procedure adopted for the tava and flat plate cooking unit. The various flow and geometric conditions used in the computational analysis are also explained in this section.

4.1 STATEMENT OF PROBLEM

The first physical model considered for the present analysis is a bowl type unit (tava), which consists of a double walled annular portion through which the HTF is circulated during the cooking period. This unit has an inner bowl of dimensions 290 mm diameter and 105 mm depth, and an outer bowl of 350 mm diameter and 135 mm depth, with an annular gap of 30 mm between the two. Further, it is considered that the edible oil (olive oil)
kept in the inner bowl is heated by circulating the hot HTF through the annulus during cooking. This unit has an axially placed inlet at the bottom, and the outlets are located horizontally at a vertical depth of 60 mm from the top. The analysis is carried out by varying the number of outlets, various heat transfer coefficients ($h_o$), and also under different HTF conditions at the tava inlet. The isometric view of the physical model, having one outlet showing the dimensions is given in Figure 4.1. The two and four outlets of the tava models are shown in Figure 4.2.

Figure 4.1 Solid model of tava cooking unit

Figure 4.2 Isometric view of the tava (a) two and (b) four outlet model
The second physical model considered for the present analysis is a hollow box configuration made of 5 mm thick mild steel plates with 45 mm spacing between the top and bottom plates having a dimension of 400 x 400 mm. A circular center portion having 200 mm diameter is considered in contact with the food being cooked and the remaining portion is exposed to the ambient. The analysis is carried out for different HTF conditions at the plate inlet, and also for various heat transfer coefficients. The solid model of the flat plate configuration is shown in Figure 4.3.

![Figure 4.3 Solid model of flat plate cooking unit](image)

4.2 MODELING AND MESHING

In the tava case, the HTF domain model consists of the annular portion of the cooking unit (the region between the top and inner bowl) along with the fluid flow region of the inlet and outlet pipes. The model is created using Pro-E software, and this model is meshed with a tetrahedral element, using hypermesh software as shown in Figure 4.4, and this meshed model is exported to the Fluent software for analysis.
Figure 4.4 Computational grid of a tava unit

In the flat plate case, the HTF domain model consists of the annular portion of the cooking unit (the region between the top and bottom plate). The top plate of the cooking unit is having 5 mm thickness which is modeled as solid domain in CFD to incorporate the conjugate heat transfer model. The CAD model is created using Pro-E software, and this model is meshed with a tetrahedral element, using hypermesh software as shown in Figure 4.5, and this meshed model is exported to the Fluent software for analysis.

Figure 4.5 Computational grid of a flat plate unit
4.3 COMPUTATIONAL PROCEDURE

The following assumptions are made for the CFD analysis:

i) The density, specific heat and thermal conductivity are considered to be constant for the HTF.

ii) The outer surface of the cooking unit in the case of tava unit, and side and bottom side in the case of flat plate unit are perfectly insulated, and hence, the heat flux is considered as zero.

The mathematical equation that governs the physical phenomenon of the incompressible flow through the annular portion of the tava and flat plate cooking unit, is given below:

Continuity equation: \( \nabla \cdot (\bar{\varepsilon}) = 0 \)

Momentum equation:

\[
\nabla \left( \rho \bar{v} \bar{v} \right) - \nabla p + \nabla \left( \tau \right) = 0
\]

Convection Pressure force Viscous force

Energy equation:

\[
\nabla \left( \bar{\varepsilon} (\rho c T) \right) - \nabla \left( k_{eff} \nabla T \right) + \phi = 0
\]

Convection Molecular heat transport Viscous dissipation

The flow through the annular portion of the cooking unit is considered as turbulent and the standard \( k-\varepsilon \) model is chosen for the analysis.

4.3.1 Boundary Conditions

The various boundary conditions used in the tava and flat plate configuration are given below:
1. Inlet and outlet conditions
   - **HTF inlet**
     - Velocity: Specified velocity, normal to boundary
     - Temperature: Specified temperature
   - **HTF outlet**
     - Pressure outlet: $P_{atm}$
     - Temperature: $T_{out,HTF}$ (Applicable only to the grid cells where back flow occurs)

2. Boundary surfaces
   a) **Tava unit**
      - Inner surface: Wall boundary condition with specified heat transfer coefficient ($h_o$) and free stream temperature
      - Outer surface: Wall boundary condition with no heat flux
   b) **Flat plate unit**
      - Cooking surface: Wall boundary condition with specified heat transfer coefficient ($h$) and free stream temperature
      - Non-cooking surface: Wall boundary condition with specified heat transfer coefficient ($h$) and free stream temperature
      - Side and bottom surface: Wall boundary condition with no heat flux.

For the tava unit, the outer surface is insulated and the inner surface transfers the heat by convection, with a known outer surface heat transfer coefficient $'h_o'$ into the edible oil (olive oil) that increases its temperature during the period of cooking. The analysis is performed for the following flow and geometric parametric conditions:
HTF inlet velocity - 0.5 m/s to 2.5 m/s with an increment of 0.5 m/s

Outer surface heat transfer coefficient ($h_o$) - 50, 60, 70, 1000 and 2000 W/m²K

Number of outlets - Single, two and four

For the flat plate unit, the bottom plate and side surfaces are insulated and the top plate surface transfers the heat by convection, with a known outer surface heat transfer coefficient '$h$' at the circular center portion, that increases the heat transfer rate during the period of cooking. The remaining portion of the cooking surface is considered with low heat transfer coefficient (4 W/m²K) due to the heat dissipation by natural convection to the ambient air. In the flat plate simulation analysis, an inlet HTF velocity and temperature of 0.007 m/s and the actual inlet temperature measured from the experiment at a given time are used for the analysis. Also the CFD analysis is performed for various outer surface heat transfer coefficients '$h$', for the circular center cooking portion.

4.3.2 Grid Independence Study

A separate grid independence study was carried out for optimizing the mesh size and cell count for both the cooking units.

In the case of tava unit, Figure 4.6 shows the variation of the heat transfer rate with respect to the mesh count. It was observed 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 parametric analyses. In the case of flat plate unit, after a mesh refinement over the cooking surface, the mesh count was found to be 442,000 tetrahedral elements for fluid domain and 148,000...
structured penta elements for the solid domain to simulate conductive heat transfer.

![Figure 4.6 Variation of the heat transfer rate with respect to the grid size](image)

**Figure 4.6** Variation of the heat transfer rate with respect to the grid size

![Figure 4.7 Convergence history of the CFD equation for the flat plate](image)

**Figure 4.7** Convergence history of the CFD equation for the flat plate

For both the tava and flat plate cooking units, the fully implicit method is adopted with the SIMPLE algorithm as the solver option for the steady flow simulation, and the first order upwind scheme was used for solving all the conservation and turbulence equations. In all the simulation trials, a convergence criterion of $1.0 \times 10^{-4}$ is used for the mass and momentum, and $1.0 \times 10^{-7}$ is used for the energy residuals. The convergence history of the CFD analysis of the flat plate unit is given in Figure 4.7.