International Journal of Advance Research in Science and Engineering
Volume No. 11, Issue No. 06, June 2022
www.ijarse.com
IJARSE
ISSN 2319 - 8354

Simulated Performance of Solar Tunnel Dryer using Computational Fluid Dynamics

Soumya Hublikar[1], Vivekanand Shikalgar[2]

[1]*Assistant Professor Department of Mechanical Engineering, ATS SBGI Miraj, Sangli 416410 (M.S.), India.*
email: hublikarsn@sbgimiraj.org

[2]*Assistant Professor Department of Mechanical Engineering, ATS SBGI Miraj, Sangli 416410 (M.S.), India.*
email: shikalgarvd@sbgimiraj.org

ABSTRACT

Computational fluid dynamics is a numerical tool that is highly accurate to simulate a very large number of applications and processes. The CFD analysis has emerged as a viable technique to provide effective and efficient design solutions. In this paper, a CFD analysis for improving temperature distribution in a solar tunnel dryer is presented. The CFD technique is used to simulate the temperature distribution inside the chamber. For this purpose, the continuity, momentum and energy equations are considered. The results obtained by CFD analysis based on a specific geometry are presented in order to improve the temperature distribution.

***Keywords**: Computational fluid dynamics,solar tunnel dryer*

1. INTRODUCTION

Drying is the oldest preservation technique of agricultural products and it is an energy intensive process. High prices and shortages of fossil fuels have increased the emphasis on using alternative renewable energy resources. Drying of agricultural products using renewable energy such as solar energy is environmental friendly and has less environmental impact.

India has been identified as one of the top twelve mega-biodiversity centers of the world with immensely rich medicinal, aromatic plants, fruits and vegetables occurring in diverse ecosystems. India which is the biggest repository of medicinal and aromatic plants in the world has to maintain an important position in the production.

The most common tunnel dryer is the Hohenheim type developed 20 years ago for tropical countries. The dryer has three main components: the collector; the dryer and the fan (optional). The collector is an air heater. It includes an absorber (black-metallic surface) that warms up the air by convection and a clear cover (UV-stabilized polythene) that warms up the air by radiation and reduces heat loss from the absorber. The dryer is where the product is spread on mesh trays (lateral loading); the structure is the same as for the collector without the absorber. An optional fan (forced convection) can be included that forces the air through the dryer. The fans can be driven by mains current, a 12V photovoltaic solar module or a car battery (Gnanaranjan et al., 1997). The choice of the materials used for constructing the dryer depends on what is locally available and its cost: for the building of the structure: bricks, cement, wood, plywood, slate, stone and loam; for the heat insulation at the bottom of the collector: polyurethane, cellulose, fibre, glass, wood, stone wool, cork, linen. The floor of the drying chamber is covered with sealing foil and mats (bamboo or plastic). A lower cost small scale model was developed for smallholder farmers. The new models worked either with alternate current or solar photovoltaic to activate fans.

The specific objectives of this study are

- Design of a tunnel type solar drying system for leafy vegetables.
- To analyse the temperature and air flow distribution inside solar dryer house by using three-dimensional computational fluids dynamic as a simulation tool.

2. METHODOLOGY

Computational fluid dynamics is a tool that can be used to analyze the solar tunnel dryer design before the actual dryer is build. Although physical experimentation allow for accurate environment measurement without the need for modeling assumption, it would not only require expensive equipment but also required large amount of time.Therefore for 1 kg of leafy vegetable per batch, CFD design simulation has been done before the actual structure is developed.

2.1 Computational fluid dynamics Simulation

In this work three-dimensional simulations have been carried out by means of commercial CFD software. ANSYS 13 FLUENT is one of the most widely used commercial codes for simulating engineering fluid flow due to its accuracy, robustness and convenience. ANSYS 13 FLUENT solvers are based on the finite volume method. The fluid region is decomposed into a finite set of control volumes. General conservation (transport) equations for mass, momentum, energy, species, etc. are solved.

Continuous partial differential equations are discretized into a system of linear algebraic equations that can be solved on a computer. ANSYS 13 FLUENT consists of four software's module that takes geometry and meshes and passes the information required to perform a CFD analysis.

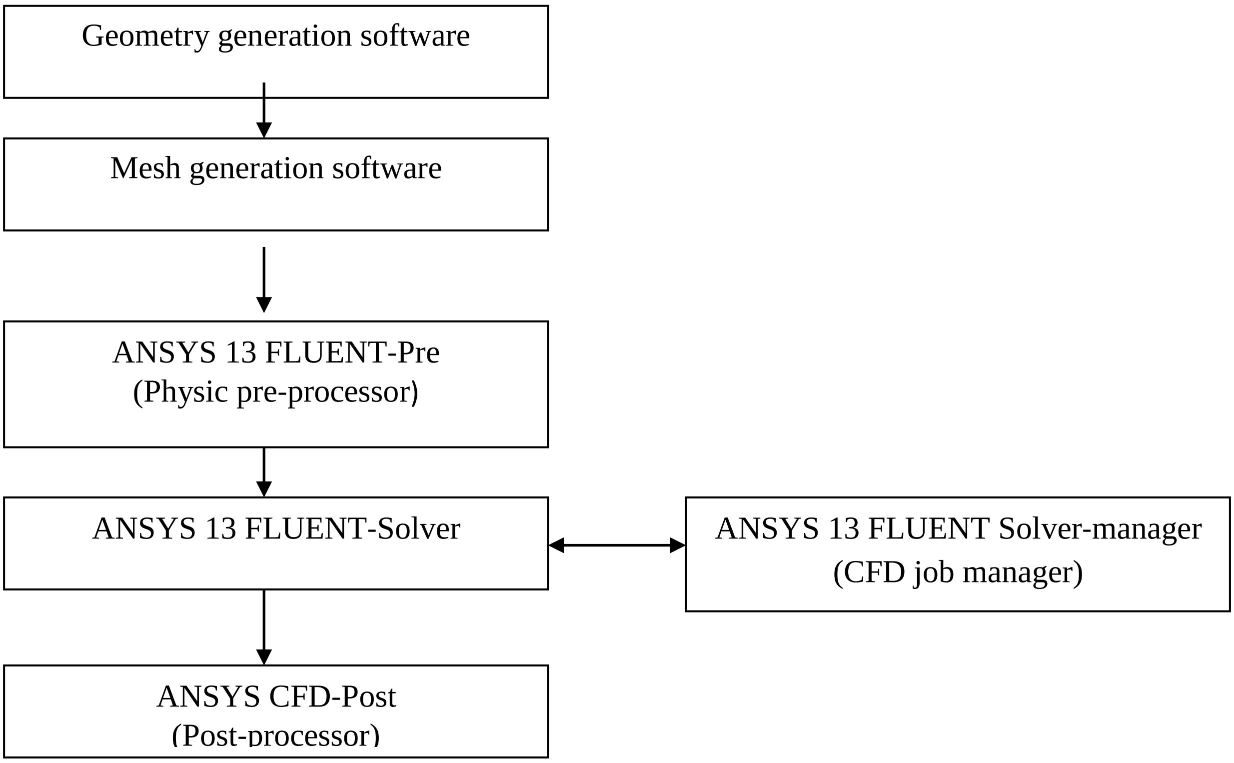

Figure 2.1: ANSYS 13 FLUENT module

2.2.1 Pre-processing stage

In the CFD simulation, pre-processing stage includes geometry creation, mesh development, physical properties set-up and the implementation of solving technique and parameters.

2.2.1.1 Geometry creation

The body about which flow is to be analyzed requires modeling. This generally involves modeling the geometry with a CAD software package. Approximations of the geometry and simplifications may be required to allow an analysis with reasonable effort.

IJARSE
ISSN 2319 - 8354

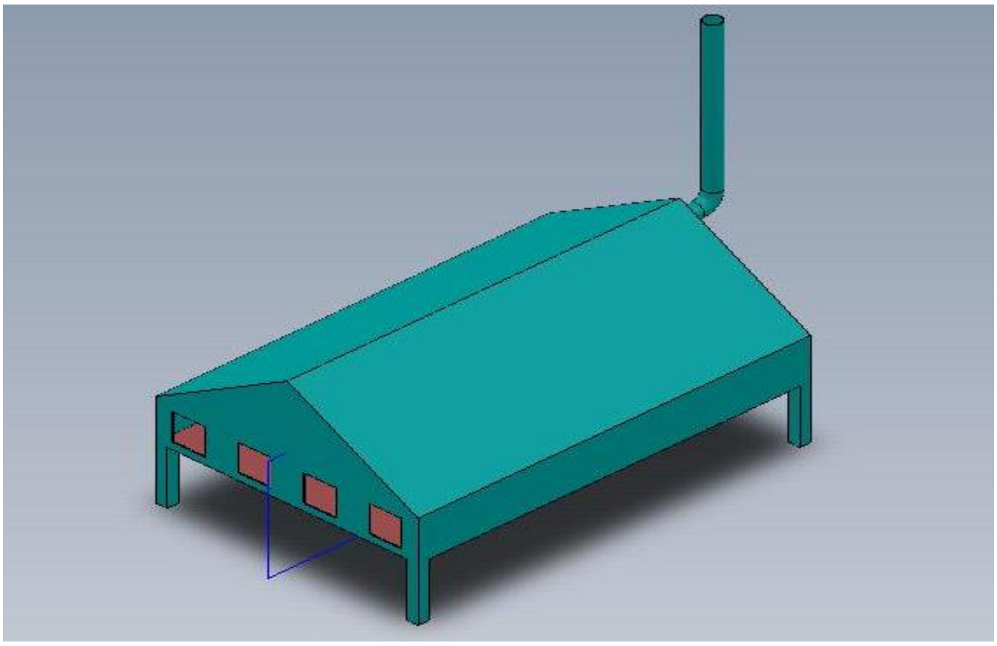

Figure 2.2: Geometric Modeling of Solar Tunnel Dryer

2.2.1.2 Mesh development

Adequate mesh resolution is important to ensure accurate result in CFD analysis.

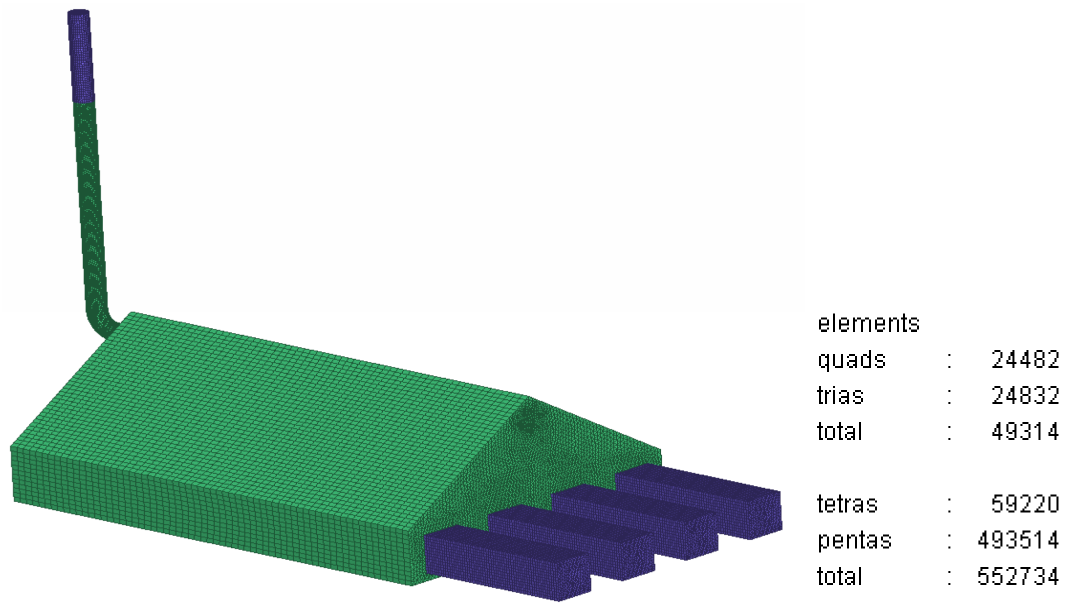

Figure 2.3: Model after the Meshing process

2.2.1.3 Physical properties set-up

The simulation is carried out for a steady state condition. Steady state condition is defined as those whose characteristics do not change with time and steady condition are assumed to have been reach after a relatively long time interval. For this simulation a physical timescale setting is used to make sure the steady state condition is achieved. For turbulence model the k-ε model has been used to represent the turbulence model for this simulation. k-ε model has proven to be stable and numerically robust and has a well-established regime of predictive capability. The k-ε model is chosen because its offers a good compromise in term of accuracy and robustness.

2.2.1.4 Boundary conditions

Boundary conditions are sets of properties or condition on a surface of a domain and are required to fully define the flow simulation. There are three boundary conditions which are the wall, inlet and outlet. In the simulation, solar radiation represents by the wall boundary condition. The solar radiation is set to be the directional radiation where the intensity value is the measured data at that particular time.

2.2.2 Solving

Once the problem physics has been identified, fluid material properties, flow physics model, and boundary conditions are set to solve using a computer.

Solver setting:

- Type: Steady Pressure based
- Turbulence model: k-ε realizable
- Fluid: Air
- Inlet: Velocity inlet
- Outlet: Pressure outlet
- Walls: Heat flux=10 w/m2
- Pressure Velocity coupling:

  Scheme- SIMPLE

  Gradient- Least Square Cell Based @ second order upwind
- Residuals: continuity, energy, k, epsilon, x-y-z-momentum (10E6)
- No of insertions: 2000

2.2.3 Post processing

In the post-processing stage, the result from the ANSYS-FLUENT solver was visualized in the following forms;

i) Contour plot for temperature distribution.

ii) Vector plot for air flow velocity and direction.

3. SIMULATION RESULTS

The temperature distribution and the velocity distribution inside the solar dryer is shown in figures 3.1 and 3.2 respectively. It could be seen that the highest temperature is 345K and observation of the velocity distribution indicated that the flow velocity in the solar dryer is not uniform.

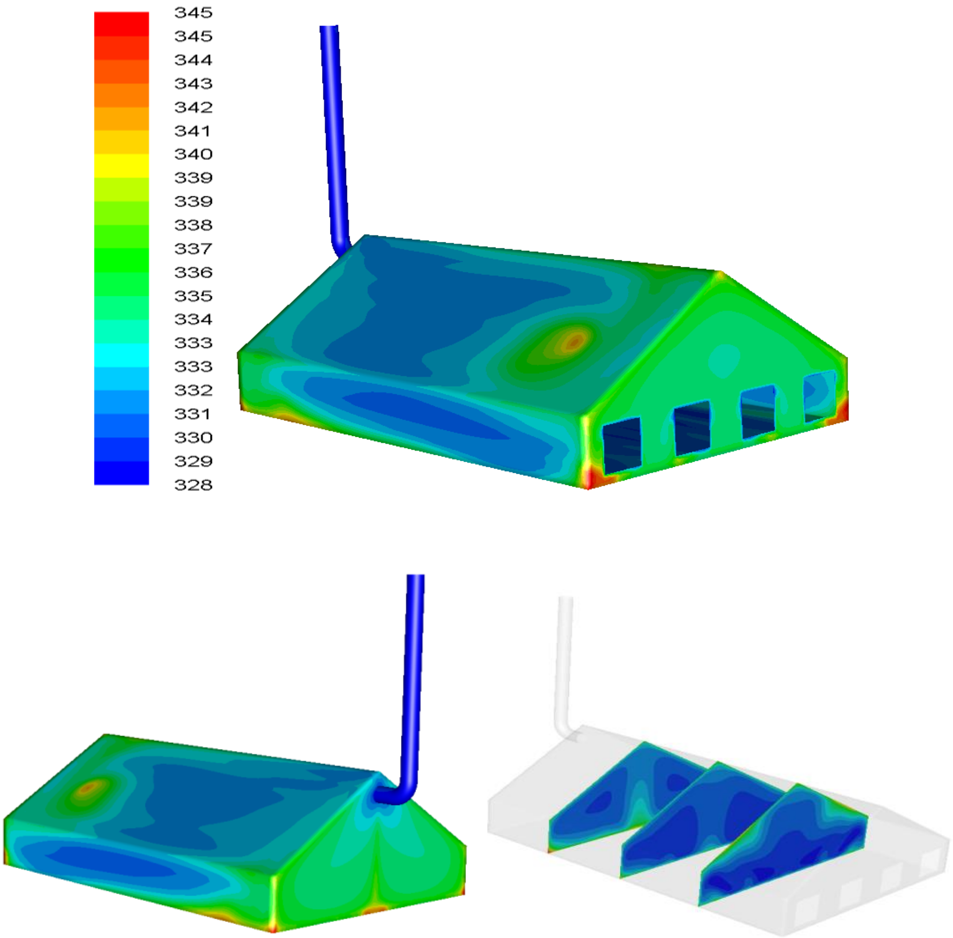

Figure 3.1: Temperature distribution from CFD simulation

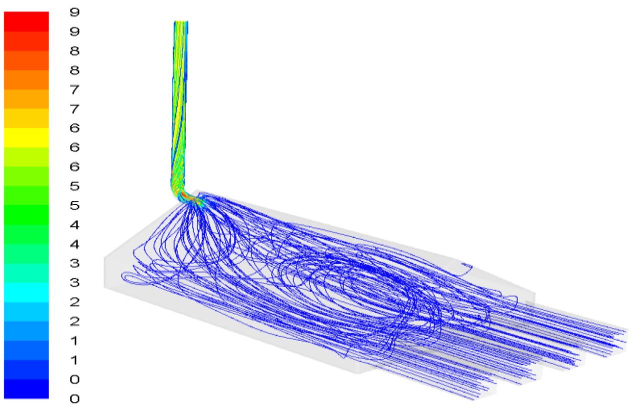

Figure 3.2: Air flow velocity profile

4. CONCLUSIONS

In this paper, a CFD analysis for improving temperature distribution in a tunnel dryer was presented. The continuity, momentum, and energy equations were solved using the CFD software. Tunnel dryers have proven to be successful in drying most agricultural produce. CFD simulation is useful in predicting air velocity and temperature profiles in a drying chamber. The commercial CFD package ANSYS-FLUENT 13 was used to predict the three dimensional flow in the tunnel dryer. Simulation in 3D provides superior results because it represents the actual problem. The result shows uniform temperature distribution of airflow throughout the dryer. The velocity of airflow was homogeneous throughout the dryer.

REFERENCES

1) Arora, S., *et al*., Convective Drying Kinetics of Red Chilies, *Drying Technology*, *24* (2006), 2, pp. 189-193
2) Gupta, P., *et al*., Drying Characteristics of Red Chilli, *Drying Technology*, *20* (2002), 10, pp. 1975-1987
3) VijayaVenkataRaman, S., *et al*., A Review of Solar Drying Technologies, *Renewable and Sustainable Energy Reviews*, *16* (2012), 5, pp. 2652-2670
4) El, A., Shalaby, S., Solar Drying of Agricultural Products: A Review, *Renewable and Sustainable Energy Reviews*, *16* (2012), 1, pp. 37-43
5) Hossain, M., *et al*., Design and Development of Solar Dryer for Chilli Drying, *International Journal of Research*, *2* (2015), 1, pp. 63-78
6) Papade, C., Boda, M., Design and Development of Indirect Type Solar Dryer with Energy Storing Material, *International Journal of Innovative Research in Advanced Engineering*, *1* (2014), 12, pp. 109-114
7) Mohanraj, M., Chandrasekar, P., Performance of a Forced Convection Solar Drier Integrated with Gravel as Heat Storage Material, *Journal of Engineering Science and Technology*, *4* (2009), 3, pp. 305-314
8) Charmongkolpradit, S., *et al*., Drying Characteristics of Chili Using Continuous Fluidized-Bed Dryer, *American Journal of Applied Sciences*, *7* (2010), 10, pp. 1300-1304
9) Cortes, E., *et al*., Feasibility Analysis of Drying Process Habanero Chili Using a Hybrid-Solar-Fluidized Bed Dryer in Yucatan, Maxico, *Journal of Energy and Power Engineering*, *7* (2013), 10, pp. 1898-908
10) Marnoto, T., *et al*., The Characteristic of Heat Pump Dehumidifier Drier in the Drying of Red Chili (*Capsicum Annum L*), *International Journal of Science and Engineering*, *3* (2012), 1, pp. 22-25
11) Umayal, A., *et al*., Performance of Evacuated Tube Collector Solar Dryer with and without Heat Sources, *Iranica Journal of Energy & Environment*, *4* (2013), 4, pp. 336-342