# Making order in the Cabinet

## Integrating CFD in the Green energy design process for food industry helps identify and fix causes for uneven drying in a Solar Cabinet Dryer.

*By Sam Mathew and Ganesh Visavale, Centre for Computational Technologies Pvt. Ltd., Pune*

*B. N. Thorat, ICT (former UDCT) Mumbai*

### Potential Solar drying process

The post-harvest losses of agricultural as well as marine products can be reduced drastically by using proper drying techniques. This is done by reducing the moisture content of the food items which makes conditions inhospitable for microbial action thereby increasing the shelf-life. The use of solar energy for this purpose has a wide scope. India receives an enormous amount of solar energy: on an average around 5 kWh/$m^2$ per day for over 300 days in a year. This can be efficiently used for drying agricultural and marine products before their safe storage. Solar drying of agricultural products in *enclosed structures* by forced convection is an attractive way of reducing post-harvest losses and low quality of dried products associated with traditional open sun-drying methods. The greatest advantage is the greater control over the drying process and ensuring hygienic drying. However, non-uniform drying of food products in such enclosed solar dryers has posed a major challenge for their commercial scale implementation.

### Solar Cabinet Dryer (SCD)

A solar cabinet dryer includes: (i) a mechanism to heat the ambient air, (ii) a drying unit where the moisture removal takes place, and (iii) an air handling unit. The present case considered a small scale solar cabinet dryer, used for drying of agricultural and marine products at ICT Mumbai which was analyzed for its flow distribution using CFD. In this SCD, the ambient air after being heated by a solar collector of size 20 sq. meters is forced by a centrifugal pump through an air channel located on the right side of the cabinet (Fig. 1). However, non-uniform drying of the product was observed. The term 'non- uniformity of drying' refers to the uneven drying of material at various locations in a dryer. Since the drying rate is a strong function of the air flow rate, a lack of flow homogeneity often contributes as the main factor for the malfunctioning of commercial scale dryers.

a) b) c)

Fig 1: Experimental Set-up – a) SCD with solar panel & centrifugal pump; b) SCD isometric view; c) SCD with two columns of trays

Thus, it was of primary importance to know the air flow pattern in the drying cabinet in order to study the areas of adequate or inadequate air velocities, which respectively lead to uneven drying. This was intended to be verified and fixed using virtual prototyping.

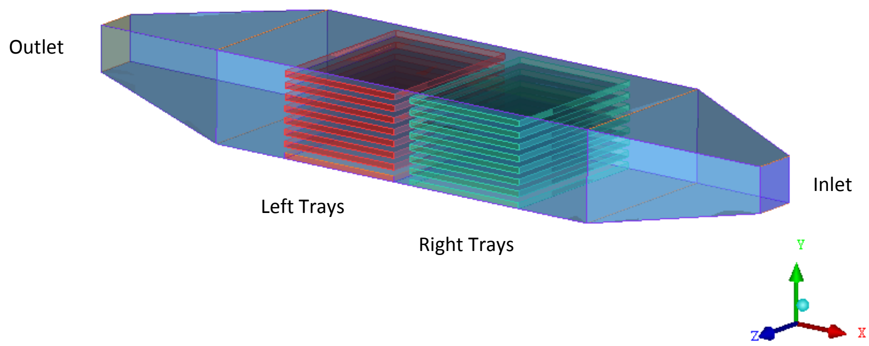


Fig 2: Computational domain of SCD in ICEM-CFD

## CFD in the design process

The flow distribution in the cabinet dryer was of interest and therefore the CFD simulation was performed on the closed domain of the drying cabinet of the SCD. A structured multi-block mesh was generated using Ansys ICEM-CFD with half-a-million cells, the mesh density being higher around the trays to capture the higher gradients. The input at the inlet of the cabinet was taken according to the profile of a regular centrifugal pump which shows a favorable flow at the outer periphery. Ansys FLUENT helps to implement experimental profile values onto the inlets through appropriate range specific UDFs and the strong post-processing features of CFX-Post help visualize the overall flow phenomenon. The simulation results confirmed the experimental observation of preferred drying of the first three rows in both the columns. The velocity vector plots (Fig 3) show a reversed flow in the bottom rows of the trays just adjacent to the inlet (Right Trays in Fig 2).

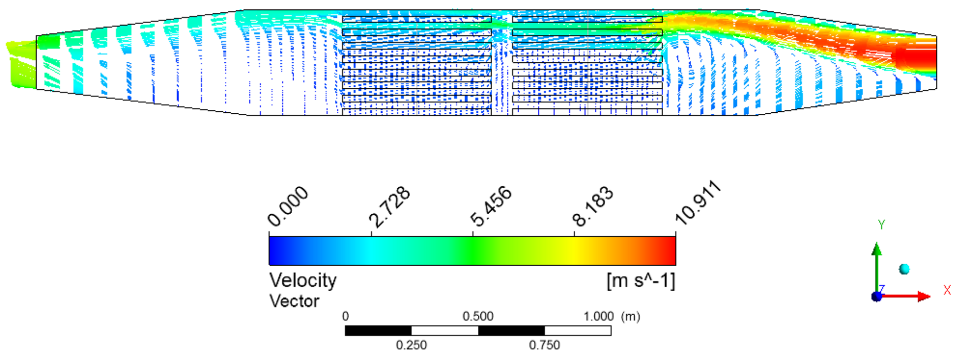


Fig 3: Flow distribution in the original SCD design

*The non-uniform inlet profile (typical for centrifugal pumps) creates large recirculation zones as it encounters the diverging section of the SCD before striking the Right Trays.*

These CFD observations were confirmed by experimental studies where the trays were filled with water and the amount of water evaporated from each tray was evaluated after regular intervals which also showed a lower evaporation rates at the bottom trays, especially the Right Trays adjacent to the inlet. The Left Trays comparatively had a more uniform flow after the air had passed through the first set of Right Trays.

The non-uniform flow was attributed to the uneven flow entering already from the inlet. This was further enhanced as the flow passed through the inlet diverging section due to *Coanda effect*. Hence, the flow clung to the top section of the cabinet.

As there was no possibility to extend the inlet region to make the flow uniform, a possible solution was to introduce a hybrid mesh-plate (with different pore sizes for the upper and lower half) after the inlet preceding the first column of Right Trays. Ansys FLUENT's capability to define porous jump boundary condition for such internal faces with appropriate model parameters helped model the mesh-plate. A parametric study helped locate the appropriate position of the mesh plate to be half-way between the inlet and the end of the diverging section. Introducing the mesh-plate meant a 25 % rise in pressure drop which would increase the load on the centrifugal pump to maintain the same mass flow rate. The end result is nevertheless favorable to ensure uniform flow over all the trays enhancing the efficiency of the batch SCD for potential commercial applications.

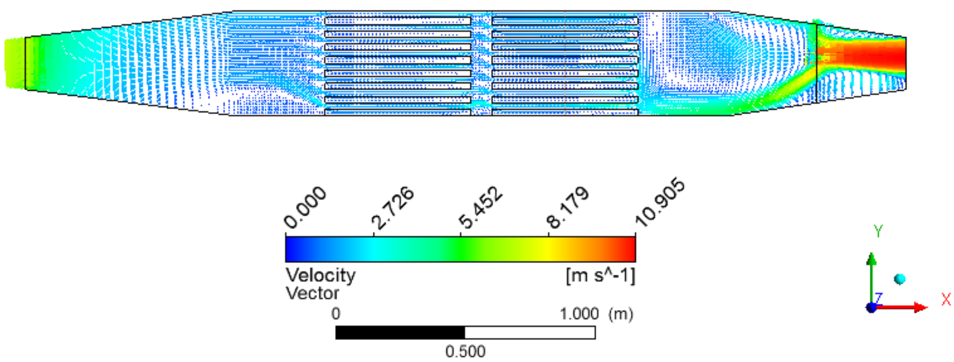


Fig 4: Flow distribution in the modified SCD design with mesh plate in the inlet section

*The top region of the mesh plate offers greater resistance to the incoming flow profile from the centrifugal pump which makes the overall flow arriving at the Right Trays to be more or less uniform.*

## Acknowledgement

The authors are grateful to Rajiv Gandhi Commission for Science and Technology for the financial support in experimentation at ICT, Mumbai. All simulation activities were carried out at Centre for Computational Technologies (CCTech), Pune.

## References:

[1] Visavale, G.L., *Ph.D. (Tech) Thesis,* Chemical Engineering Department, Institute of Chemical Technology, Mumbai, 2009.

[2] Mathioulakis, E.; Karathanos, V. T.; Belessiotis V. G., "Simulation of Air Movement in a Dryer by Computational Fluid Dynamics: Application for the Drying of Fruits," *Journal of Food Engineering,* 1998, Vol. 36, pp. 183-200.

[3] Amanlou, Y.; Zomorodian A., "Applying CFD for Designing a New Fruit Cabinet Dryer," *Journal of Food Engineering,* 2010 *(in press)*.