IJRAME

ISSN (ONLINE): 2321-3051

INTERNATIONAL JOURNAL OF RESEARCH IN AERONAUTICAL AND MECHANICAL ENGINEERING

A CFD ANALYSIS OF AIR FLOW IN A STATIONARY DRUM PARTIALLY FILLED WITH SOLID MATERIAL

Sandeep Lutade [1], Dr. S.B. Thombre [2]

[1]*M.tech, Heat Power Engg, VNIT, Nagpur, INDIA*
[2] *Professor VNIT, Nagpur, INDIA.*

Abstract

This project deals with the computational fluid dynamics analysis of air flow in stationary drum partially filled with solid material. This involves with the three dimensional analysis of air flow through a drum having tangential inlet and axial outlet. The software and analysis are to be carried out by ANSYSFLUENT. ANSYS FLUENT is Computational Fluid Dynamic (CFD) software in which flow fields and other physics are calculated in detail for various engineering applications. The analysis was done to analyze velocity and temperature distribution, pressure, air flow and turbulence. The models are first generated and then are meshed and then various velocity and pressure contours are to be drawn and graphed in this paper to analyze the air flow through drum. Standard k - ε turbulence model is allowed to predict the three-dimensional flow and the conjugate various profiles.

Keywords: Tangential Inlet, Axial Outlet, ANSYS FLUENT

1. Introduction

1.1 INTRODUCTION

Drying is a common food manufacturing process. The drying rate is a strong function of air flow or air velocity. Therefore, it is of great importance to know the air flow and velocity in the drying, thus leading to know the areas of adequate air velocities for proper drying. However, air flow and air velocity are difficult to measure during operation because several sensors are needed to be placed at various directions of air flow and locations. Since there are some difficulties in modeling the complex phenomena, computational fluid dynamic is a powerful tool to aid the prediction of drying process.

Computational fluid dynamic has been used to predict the air flow and velocity during drying.

1.2 METHODS OF DRYING:

Drying methods have been evolved around every products specific requirement. The process takes many forms and uses many different kinds of equipment. In general, drying is performed by two basic methods:

(a) Adiabatic processes

(b) Non-adiabatic processes

In adiabatic processes, the heat of vaporization is supplied by the sensible heat of air in contact with material to be dried.

In non-adiabatic processes, the heat of evaporation is supplied by radiant heat or heat transfer through walls in contact with the material to be dried.

In all drying methods, the products must be brought in contact with a medium, which is often air, in order to remove the moisture from the products surface and its surroundings.

1) Solar drying:

Solar driers operate by raising the temperature of the air to between 10-30ºC above room temperature. This makes the air move through the drier and also reduces its humidity.

There are advantages to solar drying as follows:

1. Food is enclosed in the drier and therefore protected from dust, insects, birds and animals.
2. The higher temperature deters insects and the faster drying rate reduces the risk of spoilage by micro-organisms.
3. The higher drying rate also gives a higher throughput of food and hence a smaller drying area.
4. The driers are water proof and the food does not therefore need to be moved when it rains.
5. Driers can be constructed from locally available materials and are relatively low cost.

2) Drum Dryers:

In drum dryers, the surface of a pair of rotating hot drums is coated with liquid or semi liquid form of the product. Drums are heated by steam or by direct firing inside the drum. The drums rotate slowly and the over the course of about 300^0 of rotations the product is dried. The product is scraped off with blades in flakes or sheets.

Vol.1 Issue.3, July 2013. Pgs: 1-16

3) Vacuum Dryer:

In vacuum drying, the boiling point of water is lowered below 100^0 C by reducing the pressure. If the atmospheric pressure is reduced 100 times, then boiling point will be around 0^0 C. The degree of vacuum and the temperature for drying depend on the sensitivity of the material to drying rate and the temperature.

4) Microwave Dryers:

In microwave drying, the product is exposed to very high frequency electromagnetic waves. The transfer of these waves to the product is similar to the transfer of radiant heat. As a result of high frequency waves, water molecules are polarized and tend to change orientation. In the process of orientation, sufficient heat to expel moisture from the product is generated.

5) Spray Dryers:

Spray dryer are used for dehydrating fluids. The fluid is introduced in the heated air stream in spray form. Dried product is separated from the air stream and is collected for the further processing. The design of spray dryer ranges from very simple to very complex, depending upon the fluid.

6) Cabinet Dryers:

A cabinet dryer can be a small batch tray dryer. Heat from the drying medium to the food products is transferred by convection. The convention current passes over the products, not through the products. It is suitable for dehydration of fruits, vegetables and meat and its products. The main feature of a cabinet dryer is its small size and versatility.

7) Conveyor Band Dryers:

A conveyor band dryer consists of single or several conveyor bands as the main component. The wet material is fed evenly at the feed end and is conveyed along the length of the dryer. As in a belt trough dryer, hot air is forced through the bed of moving material. However, unlike the belt trough dryer air is not forced at high rates to support wet material. The dried product is continuously discharged at the end of the dryer.

2.1 Literature Review:

Computational fluid dynamics (CFD) has been used to predict the air flow and velocity during drying. Mathioulakis et al. (1998) used CFD to simulate the air movement inside an industrial batch-type, tray air drier. Drying tests of several fruits were performed and the result showed that the degree of fruit dryness depended on its position within the drier. Determination of pressure profiles and air velocities by CFD showed that the main cause of the variations in drying rates and moisture contents was the lack of spatial homogeneity of air velocities within the drier. With the aid of CFD, Mirade and Daudin (2000) studied velocity fields in a modern sausage drier in order to provide information on air circulation inside the drier, which showed that CFD was able to predict the effects of filling level on air-flow patterns and also to identify measurement errors in areas where the main air flow direction was horizontal.

Although, the flow pattern and air velocity in the drier can be predicted using CFD modelling, further study on how to control the drying process and to reduce the energy cost is still a research topic for CFD modelling, but there are no studies have been reported so far on CFD Analysis of air flow in stationary drum partially filled with solid material.

The aim of this thesis is to propose three dimensional CFD analysis of air flow in a stationary drum partially filled solid material and predict the distribution of the inside the drum and calculation of time required for drying and capacity of blower.

3. Objectives of Current Research:

The objective of the present study is to provide more complete understanding of hot air Flow in a stationary drum and to carry out the velocity, temperature and pressure profiles and time required for drying.

A SIMPLE scheme is used with the aid of the computational fluid dynamics (CFD) ANSYS FLUENT. The governing equations for the energy and momentum conservation were solved numerically with the assumption of three-dimensional steady flow. An effective model, the standard based k-ε turbulence model was applied in this investigation. Before doing the analysis it is important to have an overview of what fluent is and how does it work.

4. Introduction to CFD:

4.1 What Is CFD?

CFD or computational fluid dynamics is predicting what will happen, quantitatively, when fluids flow, often with the complications of simultaneous flow of heat, mass transfer (e.g. perspiration, dissolution), phase change (e.g. melting, freezing, boiling), chemical reaction (e.g. combustion, rusting), mechanical movement (e.g. of pistons, fans, rudders), stresses in and displacement of immersed or surrounding solids.

Computational fluid dynamics (CFD) is one of the branches of mechanics that uses numerical methods and algorithms to solve and analyze problems that involve fluid flows and heat transfer. Computers are used to perform the millions of calculations required to simulate the interaction of liquids and gases with surfaces defined by boundary conditions. Even with high-speed supercomputers only approximate solutions can be achieved in many cases.

4.2 Advantages of CFD:

1. There are many devices and systems that are very difficult to prototype. Often, CFD analysis shows on part of system or phenomenon happening within the system that would not otherwise be visible through any other means
2. CFD is a tool for predicating what will happen under a given set of circumstance. One provides the input variables and it provides outcomes. In a short time .one can predict how a design will perform and test many variations until an optimal result is obtained. To achieve these in physical prototyping and testing (done in past) would require a huge amount of time and labour.
3. The better and faster design or analysis leads to shorter design cycles. Time and money are saved. Products get to the market faster. Equipment improvements are built and installed with minimal down time. Thus, CFD is a tool for compressing the design and development.

4.3 How Is The Working Done In CFD?

Working in CFD is done by writing down the CFD codes. CFD codes are structured around the numerical algorithms that can be tackle fluid problems. In order to provide easy access to their solving power all commercial CFD packages include sophisticated user interfaces input problem parameters and to examine the results. Hence all codes contain three main elements:

1. Pre-processing.

2. Solver

3. Post - processing.

PRE-PROCESSING:

Pre-processor consists of input of a flow problem by means of an operator friendly interface and subsequent transformation of this input into form of suitable for the use by the solver.

The user activities at the Pre-processing stage involve:

1) Definition of the geometry of the region: The computational domain. Grid generation is the subdivision of the domain into a number of smaller, no overlapping sub domains (or control volumes or elements Selection of physical or chemical phenomena that need to be modelled).

2) Definition of fluid properties: Specification of appropriate boundary conditions at cells, which coincide with or touch the boundary. The solution of a flow problem (velocity, pressure, temperature etc.) is defined at nodes inside each cell. The accuracy of CFD solutions is governed by number of cells in the grid. In general, the larger numbers of cells better the solution accuracy. Both the accuracy of the solution & its cost in terms of necessary computer hardware & calculation time are dependent on the fineness of the grid. Efforts are underway to develop CFD codes

with a (self) adaptive meshing capability. Ultimately such programs will automatically refine the grid in areas of rapid variation.

SOLVER:

These are three distinct streams of numerical solutions techniques: finite difference, finite volume& finite element methods. In outline the numerical methods that form the basis of solver performs the following steps:

1) The approximation of unknown flow variables are by means of simple functions

2) Discretization by substitution of the approximation into the governing flow equations and subsequent mathematical manipulations.

POST-PROCESSING:

As in the pre-processing huge amount of development work has recently has taken place in the post processing field. Owing to increased popularity of engineering work stations, many of which has outstanding graphics capabilities, the leading CFD are now equipped with versatile data visualization tools.

These include:

1) Domain geometry & Grid display

2) Vector plots

3) Line & shaded contour plots

4) 2D & 3D surface plots

5) Particle tracking

6) View manipulation (translation, rotation, scaling etc.)

4.4 Discretization Methods in CFD:

The stability of the chosen discretization is generally established numerically rather than analytically as with simple linear problems. Special care must also be taken to ensure that the discretization handles discontinuous solutions gracefully. The Euler equations and Navier-Stokes equations both admit shocks, and contact surfaces.

Some of the discretization methods being used are:

<u>*Finite volume method*</u> *(FVM):* This is the "classical" or standard approach used most often in commercial software and research codes. The governing equations are solved on discrete control volumes. FVM recasts the PDE's (Partial Differential Equations) of the N-S equation in the conservative form and then discretize this equation. This guarantees the conservation of fluxes through a particular control volume. Though the overall solution will be conservative in nature there is no guarantee that it is the actual solution. Moreover this method is sensitive to distorted elements which can prevent convergence if such elements are in critical flow regions. This integration

approach yields a method that is inherently conservative (i.e. quantities such as density remain physically meaningful)

Finite element method (FEM): This method is popular for structural analysis of solids, but is also applicable to fluids. The FEM formulation requires, however, special care to ensure a conservative solution. The FEM formulation has been adapted for use with the Navier-Stokes equations. Although in FEM conservation has to be taken care of, it is much more stable than the FVM approach. Subsequently it is the new direction in which CFD is moving. Generally stability/robustness of the solution is better in FEM though for some cases it might take more memory than FVM methods.

Finite difference method (FDM): This method has historical importance and is simple to program. It is currently only used in few specialized codes. Modern finite difference codes make use of an embedded boundary for handling complex geometries making these codes highly efficient and accurate. Other ways to handle geometries are using overlapping-grids, where the solution is interpolated across each grid.

Boundary element method: The boundary occupied by the fluid is divided into surface mesh.

High-resolution schemes: are used where shocks or discontinuities are present.

To capture sharp changes in the solution requires the use of second or higher order numerical schemes that do not introduce spurious oscillations. This usually necessitates the application of flux limiters to ensure that the solution is total variation diminishing.

5. CFD MODELLING:

5.1. Introduction:

Based on control volume method, 3-D analysis of air flow in a stationary drum partially filled with solid material is done on fluent software.

5.2. Geometry of model:

The geometry of the model is done using workbench.

Specification of Drum:

Length of Drum: 1m

Diameter of Drum: 1m

Inlet Area: 0.4m * 0.2m

Tray size: 1m * 0.4m

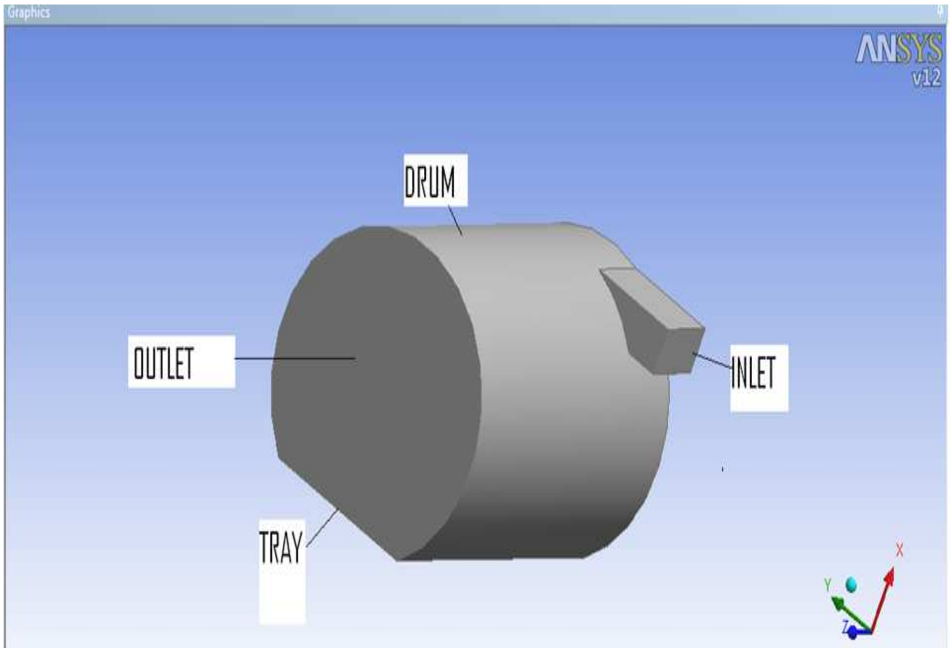

Figure 5.1 Geometry of the model in workbench

5.3 Meshing of model:

The geometry is modelled in workbench and the mesh is done to cope-up the thermal and velocity boundary layer formation as shown in figure

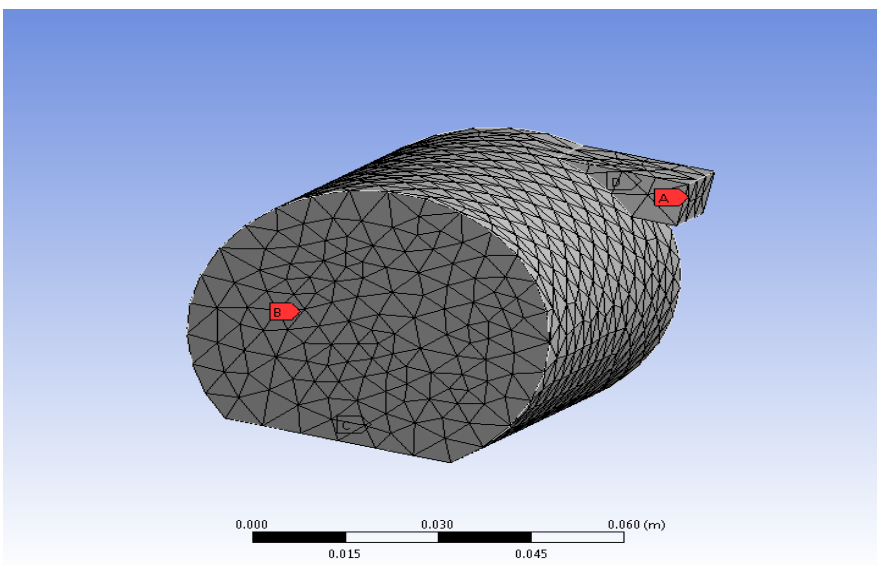

Figure 5.2 Mesh of geometry

Nodes -2235
Elements-10727

5.4 Fluent part:

A.PROBLEM SETUP

1. General

1. a. solver (i) Type-pressured based

(ii) Velocity formation-absolute

(iii) Time-steady

1. b. Model -k-epsilon

-Standard

-Standard Wall Functions

-Model constants table

Cmu	.09
C1-Epsilon	1.44
C2-Epsilon	1.92
TKE Prandtl Number	1
TDR Prandtl Number	1.3
Energy	0.85
Wall	0.85

Table 5.1 Model constants table

B.SOLUTION

1. a. Solution method (i) Scheme-Simple

(ii) Pressure-standard

(iii) Momentum-Second order

(iv) Energy- Second order

b. Solution control-

(i) Relaxation factors are taken to be default values.

Pressure	0.3
Density	1
Body force	1
Momentum	0.7
Turbulent kinetic energy	0.8
Turbulent dissipation rate	0.8
Turbulent viscosity	1
Energy	1

Table 5.2 Relaxation factors table

(ii) Convergence criterion set –

Continuity equation	10^{-3}
X-momentum equation	10^{-3}
Y-momentum equation	10^{-3}
Energy equation	10^{-6}
K equation	10^{-3}
Epsilon equation	10^{-3}

Table 5.3 Convergence criterion set

CALCULATION DONE FOR DIFFERENT RELATIVE HUMIDITY OF AIR

Table 6.1 Relative Humidity of air is taken as 10%

Sr.No	Velocity(m/S)	Discharge(m3/sec)	Time (hr)	Gr/(Re)2<< 1	Power (KW)
1	1	0.08	1.8	3.3*10^-3	10
2	1.5	0.12	1.34	1.47*10^-3	14
3	2	0.16	1.06	8.27*10^-4	19
4	2.5	0.2	0.89	5.29*10^-4	24
5	3	0.24	0.77	3.67*10^-4	29
6	3.5	0.28	0.68	2.7*10^-4	34
7	4	0.32	0.61	2.06*10^-4	39
8	4.5	0.36	0.56	1.63*10^-4	44

Table 6.2 Relative Humidity of air is taken as 50%

Sr.No	Velocity(m/S)	Discharge(m3/sec)	Time (hr)	Gr/(Re)2<< 1	Power (KW)
1	1	0.08	3.34	1.83*10^-3	10
2	1.5	0.12	2.41	8.16*10^-4	14
3	2	0.16	1.91	4.59*10^-4	19
4	2.5	0.2	1.6	2.93*10^-4	24
5	3	0.24	1.38	2.04*10^-4	29
6	3.5	0.28	1.22	1.49*10^-4	34
7	4	0.32	1.1	1.14*10^-4	39
8	4.5	0.36	1	9.07*10^-5	44

Table 6.3 Relative Humidity of air is taken as 75%

Sr.No	Velocity(m/S)	Discharge(m3/sec)	Time (hr)	Gr/(Re)2<< 1	Power (KW)
1	1	0.08	6.52	9.41*10^-4	10
2	1.5	0.12	4.71	4.71*10^-4	14
3	2	0.16	3.74	2.34*10^-4	19
4	2.5	0.2	3.13	1.56*10^-4	24
5	3	0.24	2.7	1.04*10^-4	29
6	3.5	0.28	2.39	7.66*10^-5	34
7	4	0.32	2.15	5.86*10^-5	39
8	4.5	0.36	1.96	4.63*10^-5	44

Result and Discussion:

Static Velocity:

As shown in figure, the hot air flow velocity is evenly distributed in each direction and the representative flow variables show that the residuals have stagnated and do not change with the further iteration. Therefore, the convergence in model is consistent and balance in time consuming.

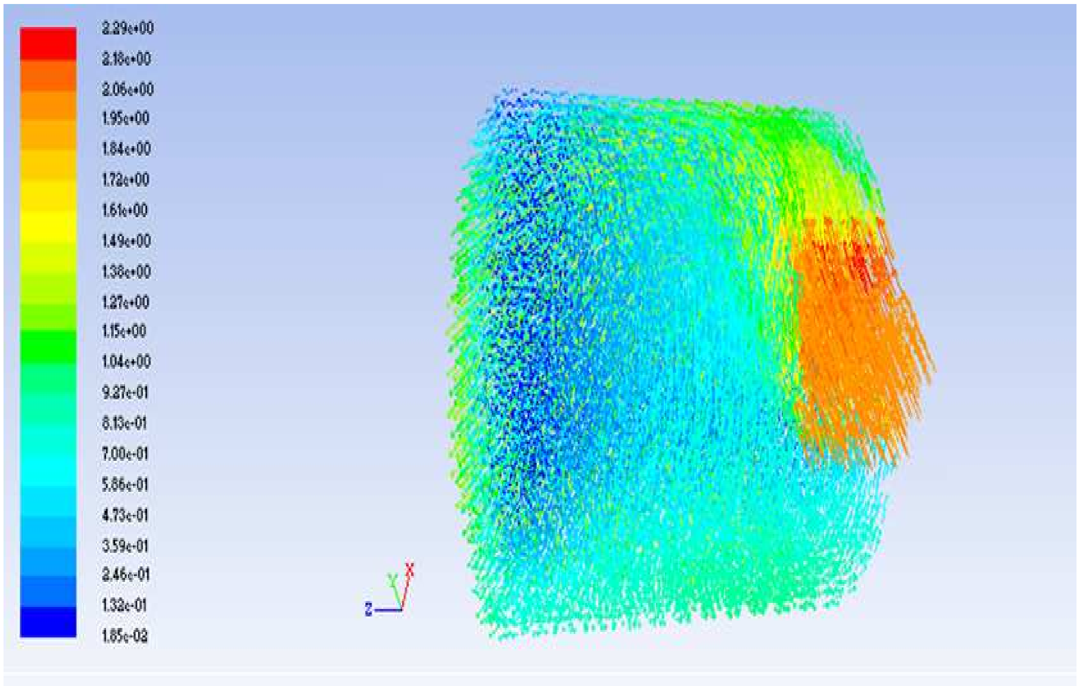

Figure 7.1 Velocity profile

Result:

Static Temperature: As show in figure, static temperature spreads the maximum temperature and distributes evenly inside the stationary drum which means that drying is efficient.

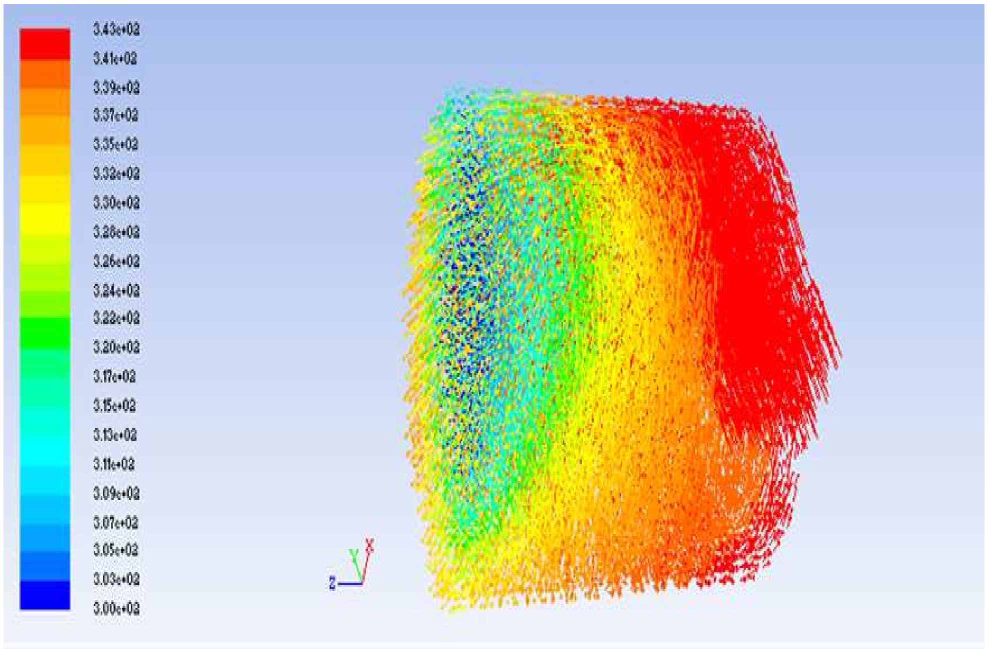

Figure 7.2 Temperature profile

Result: *Static Pressure:*

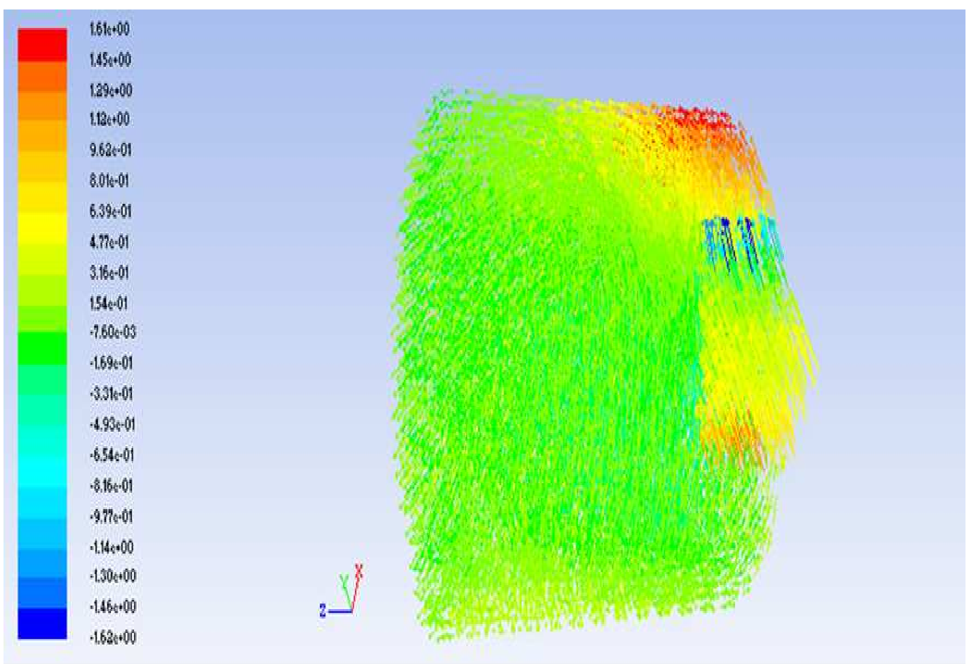

Figure 7.3 Pressure profile

Result:

Converged Solution:

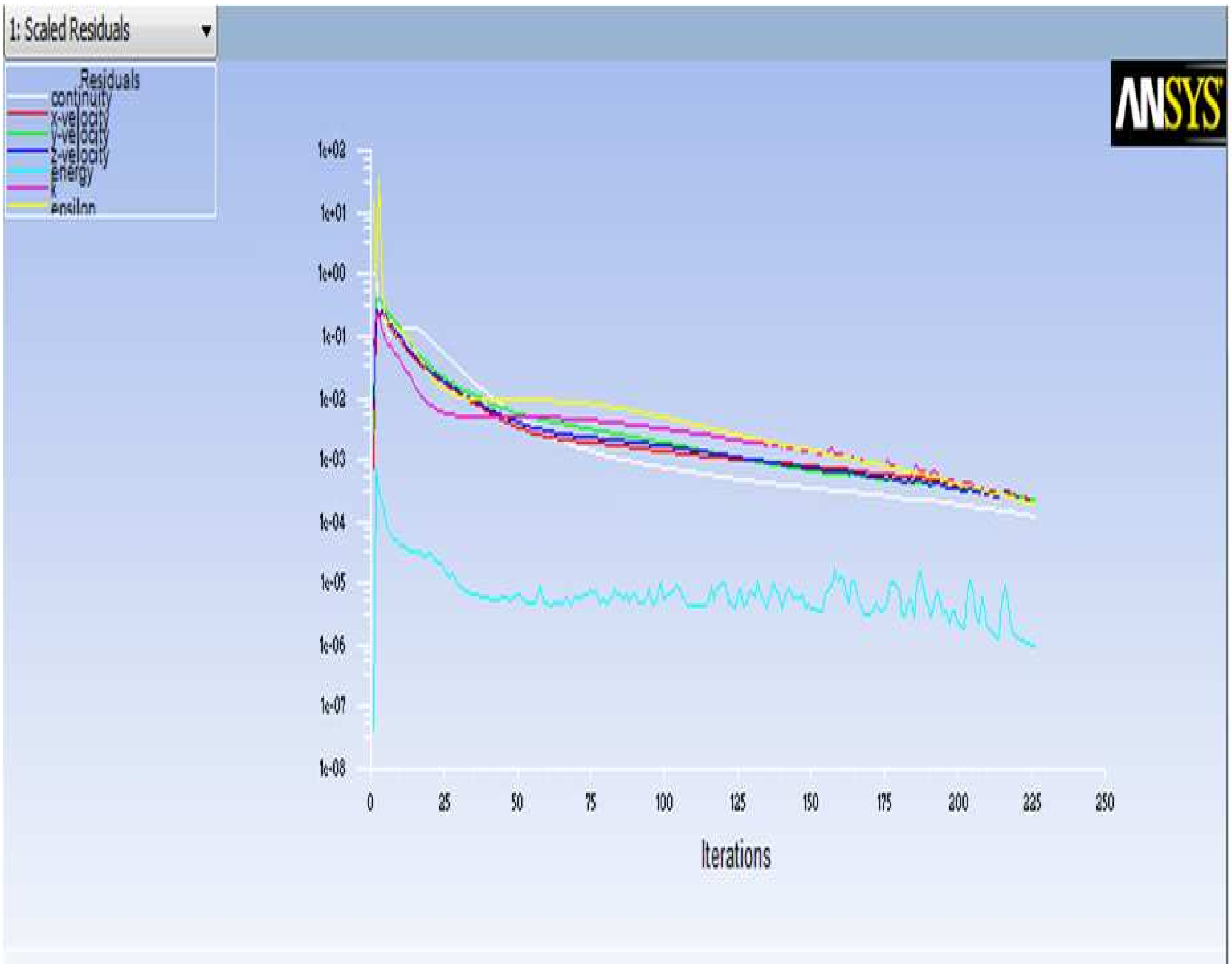

Figure 7.4 Residual plot of solution

Result:

Grahp of Time vs Velocity for different Relative Humidity of air is drawn and it is seen that as the velocity increases the time requried for the drying decreases. It is also seen that as the Relative Humidity in air increases the time also increases.

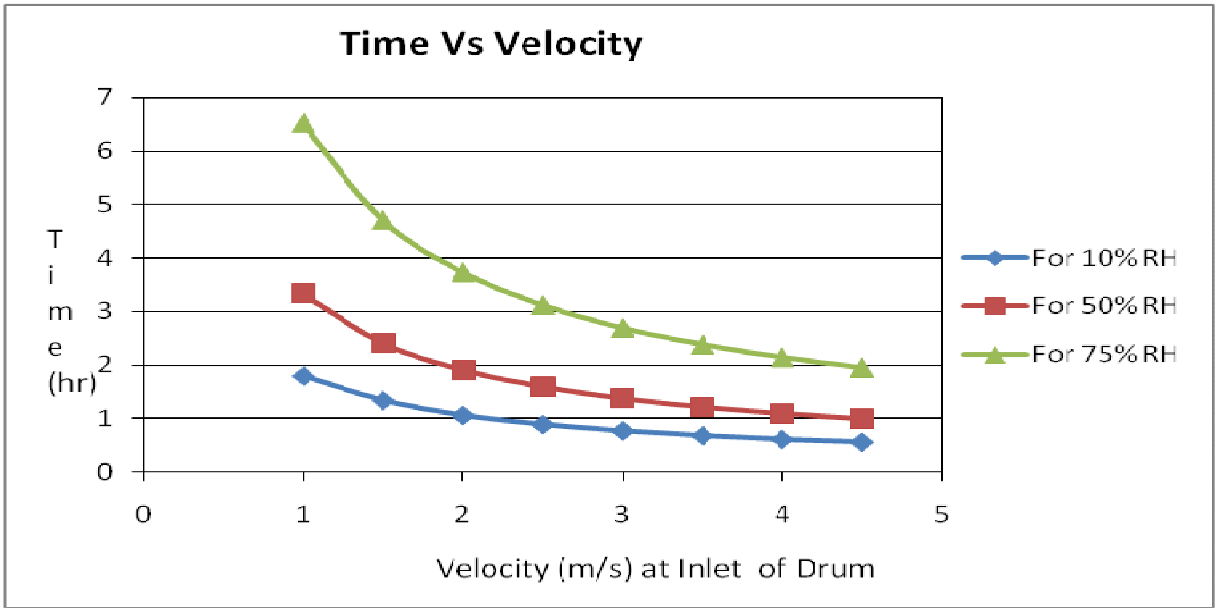

Figure 7.5 Time variation along Velocity at Inlet of drum

Result:

Graph for Time vs Power has been plot for different Relative Humidity of air to see how much capacity of hot air blower is required.It is seen that for diiferent Relative Humidity of air with power of 10 kW time required for drying increases

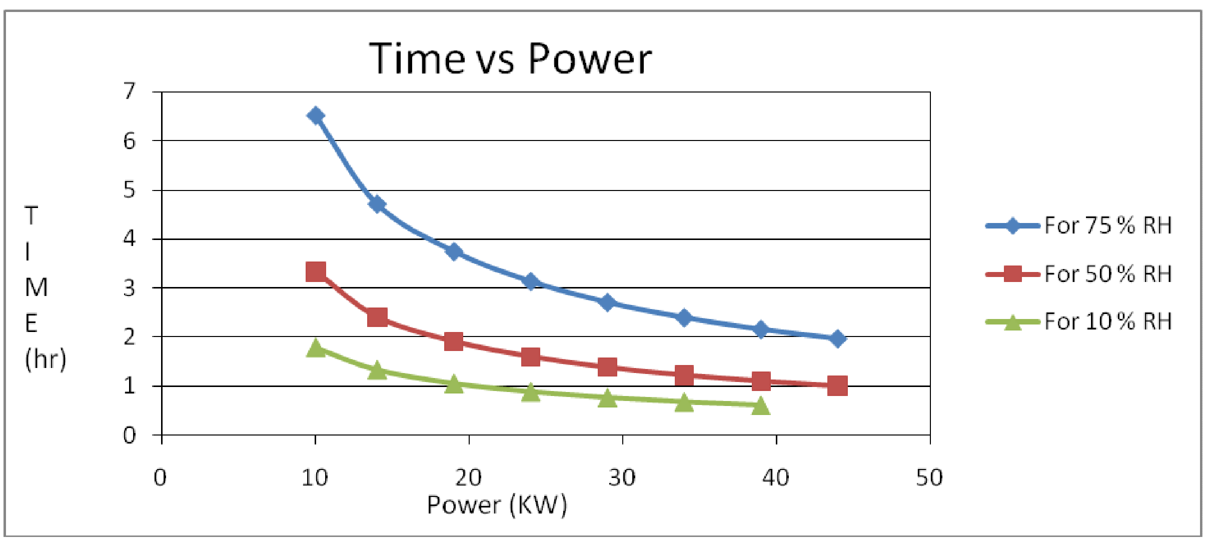

Figure 7.6 Time variation along Power

Result:

Graph for Relative Humidity vs Time has been plot and it is concluded that with velocity of 2 m/s at inlet of drum, time and power required is minimum and gives the better result.

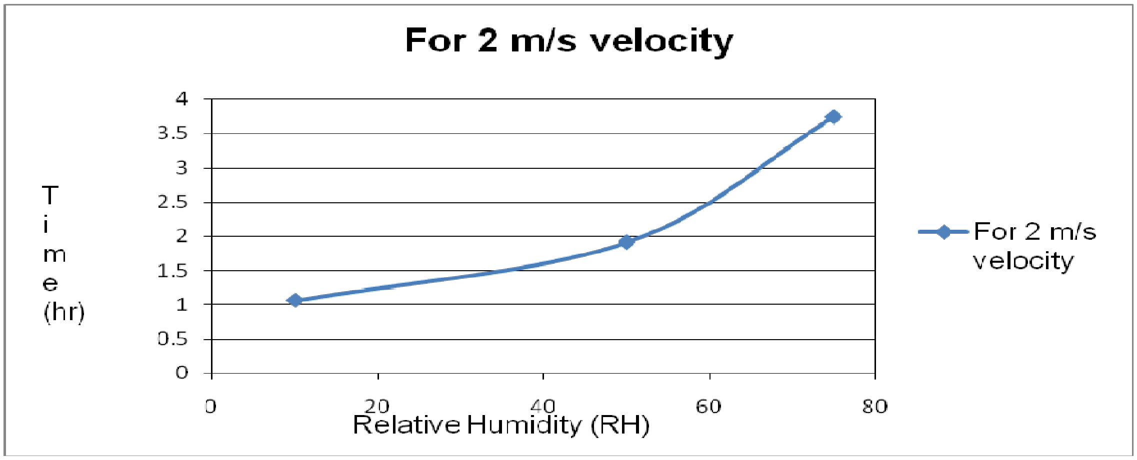

Figure 7.7 Time variation along Relative Humidity

Conclusion:

The analysis air flow in stationary drum is presented and computational fluid dynamic (CFD) for the dryer has been carried out by simulating the realistic condition to analyse the air flow distribution, temperature distribution, velocity and pressure distribution, to predict the efficiency of the dryer. The result shows that the drying system is efficient according to the measured parameter.

It is also concluded that with 2 m/s velocity at inlet of drum gives more efficient result of drying of food products and power required for it.

References:

1. ANSYS 10.0 Workbench Tutorial.
2. Patankar Suhas V, "Numerical Heat Transfer and Fluid Flow", Hemisphere Publishing Corporation, 1980.
3. J.P. Holeman, "Heat Transfer", McGraw Hill Book Company, 3rd Edition, 2002.
4. Klaus A. Hoffmann, Steve T.Chiang, "Computational Fluid Dynamics" Volume I, Engineering Education System, 1993.
5. Er. R. K. Rajput, "Heat and Mass Transfer", S. Chand, 2001.
6. Arun S.Mujumdarp, "Handbook of Industrial Drying", Third Edition edited, CRC Press, 2006.