FLOW ANALYSIS OF STENTER MACHINE

By

ADITYA SINGH

11MMET01

DEPARTMENT OF MECHANICAL ENGINEERING

AHMEDABAD-382481

May 2013

FLOW ANALYSIS OF STENTER MACHINE

Major Project Report

Submitted in partial fulfillment of the requirements

For the Degree of

Master of Technology in Mechanical Engineering (Thermal Engineering)

By

ADITYA SINGH

(11MMET01)

Guided By

Dr. R.N.PATEL

DEPARTMENT OF MECHANICAL ENGINEERING

AHMEDABAD-382481

May 2013

Declaration

This is to certify that

1. The thesis comprises my original work towards the degree of Master of Technology in Thermal Engineering at Nirma University and has not been submitted elsewhere for a degree or diploma.
2. Due acknowledgement has been made in the text to all other material used.

ADITYA SINGH

11MMET01

Undertaking for Originality of the Work

I, Aditya Singh, Roll No.11MMET01, give undertaking that the Major Project entitled "**Flow Analysis of Stenter Machine**" submitted by me, towards the partial fulfillment of requirements for degree of Master technology in Mechanical Engineering (Thermal Engineering) of Nirma university,Ahmedabad,is the original work carried out by me and I give assurance that no attempt of plagiarism has been made. I understand that in the event of any similarity found subsequently with any published work or any dissertation work elsewhere; it will resul in severe disciplinary action.

Signature of student

Date

Place:NU, Ahmedabad

Endorsed by

(Signature of Guide)

Certificate

This is to certify that the Major Project Report entitled "FLOW ANALYSIS OF STENTER MACHINE" submitted by **Mr. ADITYA SINGH (11MMET01)**, towards the partial fulfillment of the requirements for the award of Degree of **Master of Technology in Mechanical Engineering (Thermal Engineering)** of Institute of Technology, Nirma University, Ahmedabad is the record of work carried out by him under our supervision and guidance. In our opinion, the submitted work has reached a level required for being accepted for examination.The result embodied in this major project, to the best of our knowledge, has not been submitted to any other University or Institution for award of any degree.

Mr Pramod Mistry
Sr. Manager,
InspirOn Engg. Pvt Ltd.
Odhav,
Ahmedabad.

Prof S V Jain
Assistant Professor and Co-Guide ,
Department of Mechanical Engineering,
Institute of Technology, Nirma University,
Ahmedabad.

Dr R.N.Patel
Professor and Head (Guide),
Department of Mechanical Engineering,
Institute of Technology,
Nirma University,
Ahmedabad.

Dr K Kotecha
Director,
Institute of Technology,
Nirma University,
Ahmedabad.

Acknowledgments

I would like to give my special thanks to my Faculty Guide, **Dr. R.N.Patel**, Head and Professor, Department of Mechanical Engineering, Institute of Technology, Nirma University, Ahmedabad for his valuable guidance and continual encouragement throughout the project.

I would like to give my special thanks to my Industrial Guide, **Mr Pramod Mistry**, Sr. Manager, Inspiron Engg. pvt Ltd., Odhav, Ahmedabad for his valuable guidance and continual encouragement throughout the project.

I am very much thankful to **Prof. S V Jain**, Assistant Professor, (Dept. of Mech. Engg., IT, NU) who have directly or indirectly helped me during this dissertation work.

I am very much thankful to **Dr. V.J.Lakhera**, Professor, (Dept. of Mech. Engg., IT, NU) and **Dr. K Kotecha** (Director, IT, NU) who have directly or indirectly helped me during this dissertation work.

I am obliged to my family specially my parents and parents in law for their time and adjustments during the pursuit of my project. I am forever grateful to my team member **Mr. Yogesh Shah** and **Mr. Ankit Thakkar** (Design Engineer, Inspiron Engg. Pvt Ltd) for her endless support, love, humour and encouragement in my life.

Finally, I am thankful to all my friends who always motivated me throughout the course.

ADITYA SINGH

Abstract

The Stenter machine is used for the drying of woven and knitted fabrics. It is a very versatile and common machine in textile finishing. Almost every open width textile fabric is treated on Stenters during its textile processing. For maintaining universal use, the Stenter range is usually standing separately, not in a continuous line with other machines. Four types of processes are done on Stenter which are drying, heat setting, finishing and coating.

The Stenter machine usually consists of 8-10 chamber and each chamber contains 2-blowers, each blower provided in seperate casing and each casing contains 12 nozzles, i.e total 24 nozzles are provided in each chamber (12 facing down and 12 facing up). In each nozzle there is 48 openings, each of which are supplying the air to the fabric.

In present study, CFD analysis of fan house was carried out in three stages. In first stage, CFD analysis of Fan B with casing was done; in secound stage CFD analysis Fan A with casing was done; finally in third stage, CFD analysis of Fan B with nozzles were performed. Results were presented in terms of pressure and velocity contours. The CFD stimulation results were compared with experiments and CFD results provided by M/s InspirOn engg. Pvt Ltd, Odhav, Ahmedabad, and found in good agreement.

Contents

List of Figures

List of Tables

Abbreviations

Heat transfer rate (KW)	Q
Mass flow rate (kg/s)	M
Specific heat (kj/kg.K)	C_p
Area (m^2)	A
Hot fluid temperature (^0C)	Th
Cold fluid temperature (^0C)	Tc
Viscosity (kg/m.s)	μ
Heat transfer co efficient (W/m.K)	h
Watt	W
Reynold no	Re
Nusselt no	Nu
Prandtl no	Pr
Equivalent diameter (m)	De
Inside diameter (m)	di
Outside diameter (m)	do
No of tubes	Nt
Mass (Kg/s)	m
Power (KW)	p
Air flow(m^3/hr)	q
Density (kg/m^3)	ρ
Inlet and outlet diameter(m)	D1,D2
Inlet and outlet width(m)	B1,B2
Speed (Rpm)	N
No. of blades	n
Stage work	$\triangle p2$
Stage pre. Coefficient	ψ_{st}
Stage reaction	R
Stage efficiency	η_{st}
Impeller pre.coefficient	ψ_r
Mach number	M
Inlet and outlet absolute angles	α_1,α_2
Inlet and outlet relative angles	β_1,β_2
Inlet and outlet relative velocity(m/sec)	w_1,w_2
Inlet and outlet absolute velocity(m/sec)	C_1,C_2
Inspiron Engg.Pvt Limited	IEPL

Chapter 1

Introduction

A mill is a factory that houses spinning and weaving machinery, typically built between 1775 and 1930 were instrumental in the growth of the machine tool industry, enabling the construction of larger cotton mills. The requirement for water helped stimulate the construction of the canal system, and the need for power the development of steam engines.

Limited companies were developed to construct the mills, which led to the trading floors of the cotton exchange of Manchester, creating a vast commercial city. The mills also generated employment and drew workers from largely rural areas, leading to the expansion of local urban populations and the consequent need for additional housing. In response, mill towns with municipal governmentswere created.

The mills provided independent incomes for girls and women. Child labour was used in the mills, and the factory system led to organised labour. Poor conditions in cotton mills became the subject of exposés, and in England, the Factory Acts were written to regulate them.

The fabric mill was originally a Lancashire phenomenon that then was copied in New England and later in the southern states of America. In the 20th century, North West England lost its supremacy to the United States, then India and then China. In the 21st century, redundant mills have been accepted as part of a country's heritage. Cotton is the world's most important natural fiber. In 2007, the global yield was 25 million tons from 35 million hectares cultivated in more than 50 countries.

There are six stages

- Cultivating and harvesting
- Preparatory processes
- Spinning

- Weaving
- Finishing
- Marketing

The preparatory processes and spinning happen in a spinning mill, weaving happens in a weaving shed and finishing at the bleach works and dye works. Traditionally these processes occur in separate mills.

1.1 Stenter in Finishing

The Stenter is a very versatile and common machine in textile finishing. Almost every open width textile fabric is treated on Stenters during its textile Processing. For maintaining universal use, the Stenter range is usually standing separately, not in a continuous line with other machines.

1.2 Overview of Thesis

In this work, exhaustive study has been done, In Chapter 2 gives information about the Stenter machine and chapter 3 includes brief desrciption of other drying process for replacement of Stenter machine. Chapter 4 describes the calculation for heat duty , power required by the Stenter machine to dry the particular fabric and description about the blower which circulates the air in chamber. Chapter 5 includes CFD analysis of the Stenter machine to study flow analysis . Conclusion and scope of future work is given in chapter 6.

Chapter 2

Literature Review

2.1 Introduction of Stenter Machine

FUNCTION:- The Stenter range MOTEX TWIN AIR can be used for the treatment of woven And/or knitted fabrics. The Stenter itself is a very versatile and common machine in textile finishing. Almost every open width textile fabric is treated on Stenters during its textile Processing. For maintaining universal use, the Stenter range is usually standing separately, not in a continuous line with other machines. There are four type of process can be done on Stenter.

1. Drying
2. Heat Setting
3. Finishing
4. Coating .

2.1.1 Drying

The main purpose of this process is evaporating the liquid in the fabric up to certain level of residual moisture. Temperature may be reach 1200 to1900 C depending upon on fabric and desired degree of whiteness.Stenter Machine is used to do following process With knitted fabric this is also used to relax the fabric and to eliminate stresses and shrinkage in following process. Therefore the fabric width is not increased very much but you give a lot of overfeed to allow relaxation in the longitudinal direction where most stress are present.

2.1.2 Heat Setting

In this process fabric related to thermoset material like Polyester (PES) need certain temperature to set their properties which is known as Curing temperature. This temperature range is 180^0C to 210^0C.

2.1.3 Finishing

Finishing Process is to improve surface of fabric by heating process.

2.1.4 Coating

Coating is the process to add some chemical in proper manner and set them on top side of fabric to improve the properties of fabric for different application.

2.1.5 In Stenter Machine There are 6 Sections in Complete Stenter Process

1. Inlet combination
2. Padder
3. Weft Straightener
4. Stenter Infeed
5. Stenter chamber
6. Outlet combination

2.1.5.1 Inlet Combination

Inlet Combination is to prepare fabric to enter in Stenter. It consists of Tensioning and guiding system. Tensioning device create tension in the fabric for proper guiding And guiding system align the fabric with respect to machine

2.1.5.2 Padders

Padder is used for add moisture or chemical to fabric and control the moisture by squeezing the fabric. There are two type of Padder used.

- Two Bowl Padder
- Three Bowl Padder

2.1.5.3 Stenter Frame

This is main heart of Stenter machine. Stenter Process is done in this section. Stenter frame is selected on different parameters:- A. Fabric Width B. No of Chamber (min 3 and max10) C. Type Of chain D. Fabric Speed E. Heating Media

A. FABRIC WIDTH:- All the section of Stenter is depends on Fabric Width. There are 4 options to Choose.

- B180- Fabric Width 1600mm mainly used for woven
- B200- Fabric Width 1800mm mainly used for woven
- B260- Fabric Width 2400mm mainly used for Knit
- B340- Fabric Width 3200mm mainly used for Silk & Knit

B. NO OF CHAMBERS: Minimum 3 and maximum 10

C. TYPE OF CHAIN:- Chain is used to hold the fabric at both the edges width wise. There are three options to select.

- Combi (It is the combination of both)
- Pin
- Clip

D. FABRIC SPEED: Fabric speed is generally 5 to 100 m/m. (Optionally higher speed 7.5 to 150mpm or lower 2.5 to 50mpm)

E. HEATING MEDIA:-Thermic oil of direct gas heating system is used.

2.1.5.4 Entry Section

Entry Sections used to control the speed of fabric. This is also used to give Overfeed Or under feed to fabric for biaxial or axially orientation. Fabric edges are spreaded by spreading device (LA83 3finger uncurled or LS3001 mechanical spreader) to pining in the fabric on the chain at entry. The chain is conveyed the fabric through the chamber.

Entry section for Controlling speed of fabric shown Figure 2.1

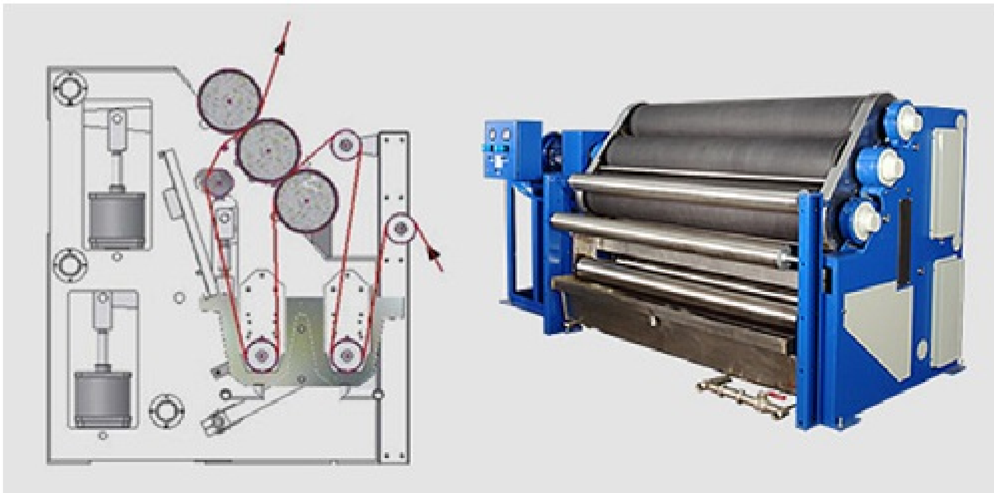

Figure 2.1: Entry of section of stenter [5]

2.1.5.5 Chamber

In this section all Stenter processes are done. In the chamber firstly fresh air is suck through a heat exchanger which is generally heated by oil or gas. Hot air is delivered by impeller and blown through the top and bottom nozzle. Generally two types of nozzles are used.

- for Woven
- for Knit

This hot air is used for Stenter process. 80% of air recirculated in chamber and only 20% air is exhausted so the same amount of fresh air sucked form Entry and exit of the chamber. Fabric width can be adjusted in the chamber during running of the chain cooling zone at the exit of the chamber to cool down the fabric to surrounding Temperature. Fabric is pin off from chain by Pin out device at exit.

Typical arrangement of important parts and direction of air-flow shown in Figure 2.2

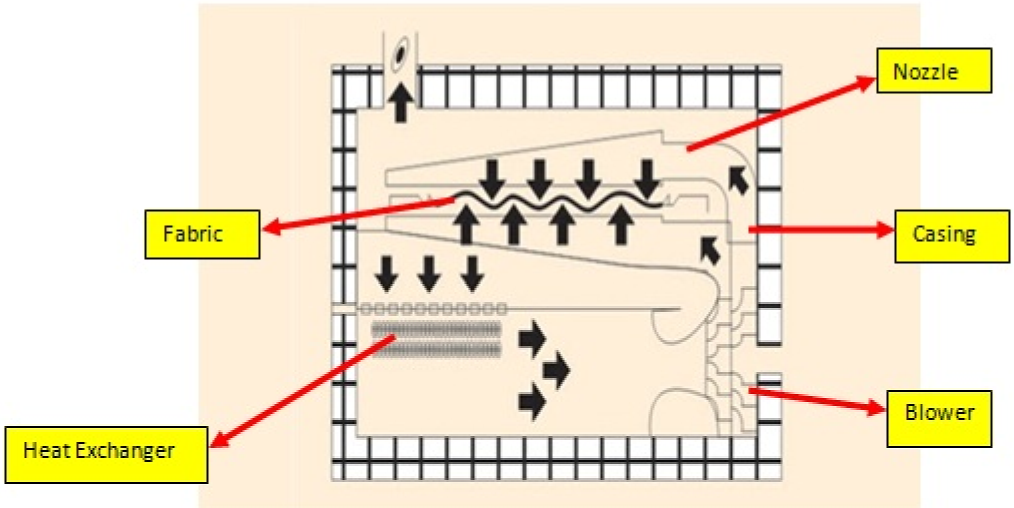

Figure 2.2: Flow process of stenter [5]

2.1.5.6 Outlet Combination

Outlet combination is used to take off of fabric from exit of the chamber and Make the fabric in batch form shown in Figure 2.3

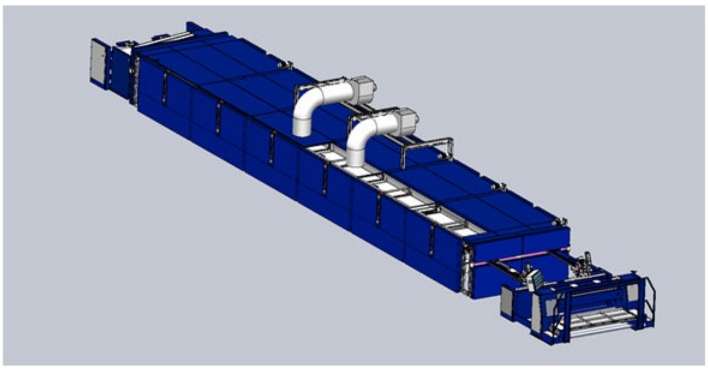

Figure 2.3: Outlet of stenter machine [5]

Chapter 3

Drying Technology

3.1 Microwave Drying For Textile

Datta and Ramaswamy[6]gives brief introduction about Microwave heating that, Microwave heating is a very promising technology which has been finding new applications in industry. It is a technology which can replace conventional heating. We would like to describe our new microwave industrial applicator used for drying textiles in manufacturing. In this drying process, a very thin layer of textile material does not have a very well defined position in the applicator. Also the complex permittivity of dried textile is not constant during the procedure. Its value changes in time with respect to the decreasing moisture content .

And also Datta and Ramaswamy[6]presents an overview of the microwave drying technology as well as it reviews the recent developments in microwave assisted drying technologies and future R&D needs in India. Recently, microwave convective and microwave vacuum drying techniques have been investigated as potential methods for obtaining high quality dehydrated food products. Microwave drying is rapid, uniform and energy efficient compared to conventional hot air drying as the microwaves penetrate to the interior of the food causing water to get heated within the food. This results in a greatly increased vapour pressure differential between the center and surface of the product, allowing rapid removal of moisture from the food.

Microwaves are electromagnetic waves having wavelength (peak to peak distance) varying from 1millimeter to 1 meter. Frequency of these microwaves lies between 0.3 GHz and 3 GHz. Microwaves have greater frequency than radio waves so they can be more tightly concentrated. Microwaves propagate through air and space at about the speed of light. Microwaves can also be considered as electromagnetic force fields for better understanding of working of microwave oven. Microwaves interfere inside the microwave oven to produce high and low energy pockets. Application of microwave energy to dry food materials is a good approach for coping with certain drawbacks of conventional drying. Microwaves penetrate to interior of the food causing water to get heated within food. This results in a greatly increased vapour pressure differential between the center and surface of the product, allowing fast transfer of moisture out of the food. Hence, microwave drying is rapid, more uniform and energy efficient compared to conventional hot air drying. The problems in microwave drying, however, include product damage caused by excessive heating due to poorly controlled heat and mass transfer .

3.1.1 Mechanism of Heating

In microwave heating or drying, microwave-emitted radiation is confined within the cavity and there is hardly heat loss by conduction or convection so that energy is mainly absorbed by a wet material placed in the cavity. Furthermore, this energy is principally absorbed by water in the material, causing temperature to raise, some water to be evaporated, and moisture level to be reduced. A domestic microwave oven works by passing microwave radiation, usually at a frequency of 2450 MHz (a wavelength of 12.24 cm), through the food. Water, fat, and sugar molecules in the food absorb energy from the microwave beam in a process called dielectric heating. Many molecules (such as water) are electric dipoles, meaning that they have a positive charge at one end and a negative charge at the other, and therefore rotate as they try to align themselves with the alternating electric field induced by the microwave beam. This molecular movement creates heat by friction as the rotating molecules hit other molecules and put them into motion.

Microwave heating is most efficient on liquid water, and much less so on fats and sugars (which have less molecular dipole moment), and frozen water (where the molecules are not free to rotate). Large industrial/commercial microwave ovens operating in the 900 MHz range also heat water and food perfectly well. The power generated in a material is proportional to the frequency of the source, the dielectric loss of the material, and the square of the field strength within it. The microwave heating rates and potential non uniformity are functions of oven factors and load characteristics (size, shape, dielectric properties, etc.).

In conventional heating, heat is transferred to the surface of the material to be heated by conduction, convection, and/or radiation, and into the interior by thermal conduction. In contrast, in dielectric heating, heat is generated directly inside the material, making possible higher heat fluxes and thus a much faster temperature rise than in conventional heating. However, heat conduction still plays an important role when heating thick samples by dielectric heating and for equilibrating temperatures when heat generation is uneven. Depending on water content the depth of initial heat deposition may be several centimeters or more with microwave ovens, in contrast to grilling, which relies on infrared radiation, or the thermal convection of a convection oven, which deposit heat shallowly at the food surface. Depth of penetration of microwaves is dependent on food composition and the frequency, with lower microwave frequencies being more penetrating. The heat generated per unit volume of material (Q) is the conversion of electromagnetic energy in to heat energy.

Method for drying the cloth by Microwave heating Shown Figure 3.1

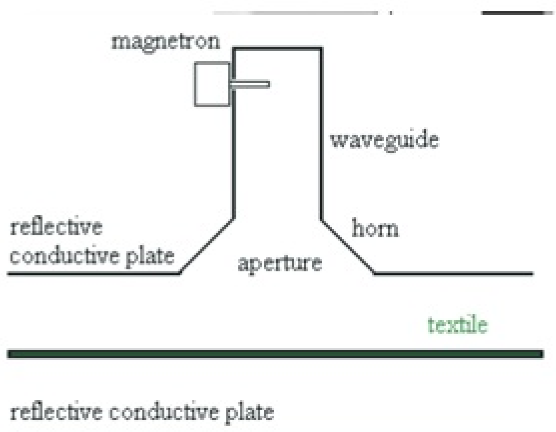

Figure 3.1: Microwave heating Process [5]

3.2 Acoustic Cloth Drying

Vladimir N. Khmelevand Igor I. Savin[9] shows result of researches, aimed to increasing the efficiency and productiveness of "classic" cloth convective drying method in drum type washing drying machine with simultaneous decreasing the power consumption by means of include the high-power acoustic radiator into washing machine design.

The integration of acoustic drying system into washing drying machine provides increasing the drying features by increasing the drying and decreasing energy consumption.

The energy consumption of drying process decreases with increasing the acoustic radiation power; 3. Maximal growth of drying speed by acoustic influence provides on initial drying period when the cloth and air into drum are cold and cloth humidity is high; 4. the most advantages of drying process with acoustic influence in comparison with drying without acoustic influence obtains when air temperature in drum is low.

Acoustic washing machine manufactured by LG Electronics shown in Figure 3.2

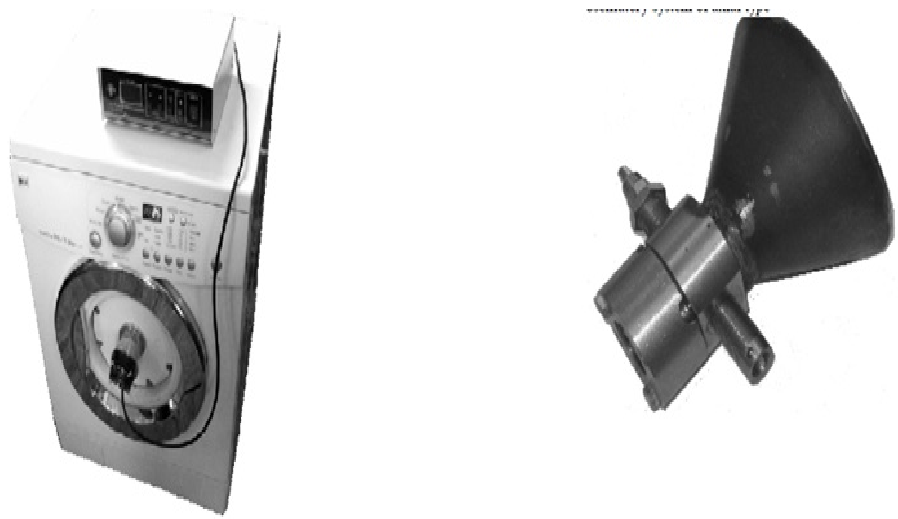

Figure 3.2: Acoustic drying taking palce in washing machine (left) Acoustic drying instrument (right) [6]

3.3 Pulse Combustion Drying Technology

In North American Waste to Energy Conference theoretical research and literature survey on pulse combustion and pulse combustion spray drying allowed us to point out the following advantages of the pulse combustion applied in drying:

•Compact drying equipment (improved heat transfer, high difference between the material and flue gas temperature, high drying rate);

• Environmental friendly operation (low emission of toxic substances, low amount of air discharged to atmosphere, efficient combustion);

• Wide variety of feedstock handled (sticky materials, heat sensitive products);

• Better atomization and handling (no need of atomizer or HP nozzle);

• Savings on auxiliary equipment (smaller motors, some of the equipment eliminated, a combustor delivers energy to run the dryer and displaces fan and requires less electrical energy).

The process of intensification comes from secondary atomization and stripping of the boundary but the effects cannot be separated due to the lack of suitable data. However, the most important factor accelerating the drying rate is definitely the high driving force of the process. The difference between gas and dried material temperatures (which may reach about 700–800K at the inlet) controls drying time and creates rapid and efficient drying process. Despite an increasing number of industrial applications and theoretical considerations the pulse combustion drying technique is still under development. Application and development of computational fluid dynamics (CFD) technique is recommended in designing of pulse combustion drying processes to avoid tiresome and costly experiments.

Pulse combustion process and pulse combustion instrument for production of hot air shown in Figure 3.3

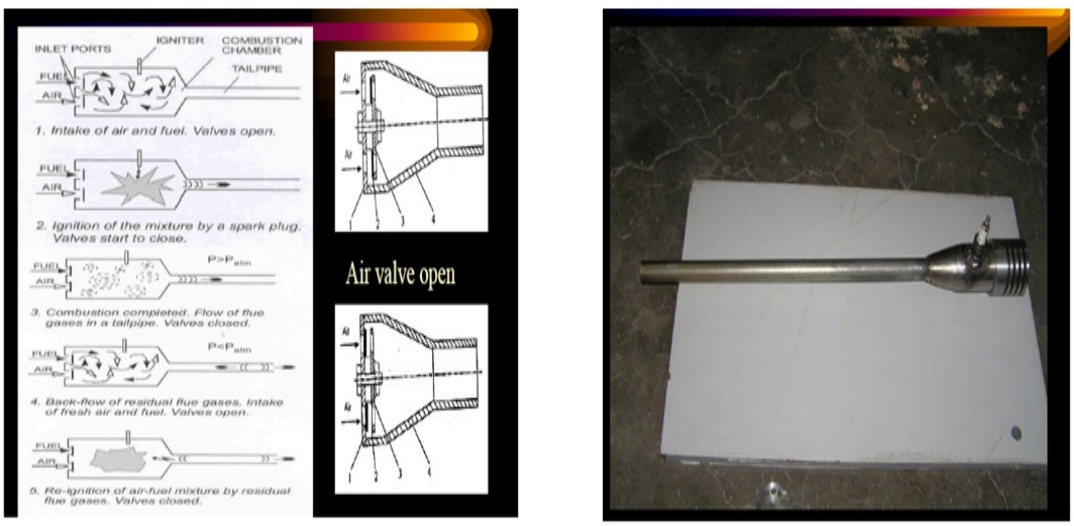

Figure 3.3: Pulse combustion process and pulse combustion instrument [4]

Comparion of drying between pulse combustion and conventional drying process shown in Table 3.1

Table 3.1: Comparison of pulse combustion with other drying process

Process Parameters	Steady State	Pulse
Combustion intensity	100-1000	10000-50000
Efficiency of burning	80-96	90-99
Loss due chemical underburning(%)	0-3	0-1
Loss due mechnical underburning(%)	0-15	0-5
Temperature level(K)	2000-2500	1500-2000
Co concentration (%)	0-2	0-1
Nox concentration(mg/m^3)	100-7000	20-70
Convective Heat transfer coefficient(W/m^2k)	50-100	100-500
Time of reaction(s)	1-10	.01-.5

3.3.1 Removal of Nox from Pulse Combustion Technology:-

Reneker and Chun;Frenot and Chronakis[9] concluded that Pulse combustion systems have many advantages when compared with conventional steady flow combustion devices, including high thermal efficiencies, low NOx emissions, and their ability to self aspirate. However, the additional complexities in designing and commissioning pulse combustion systems have severely limited their application in industrial and domestic combustion markets.

For removing Nox from pulse combustion , Tailpipe of pulse combustion dipped in water tank shown Figure 3.4

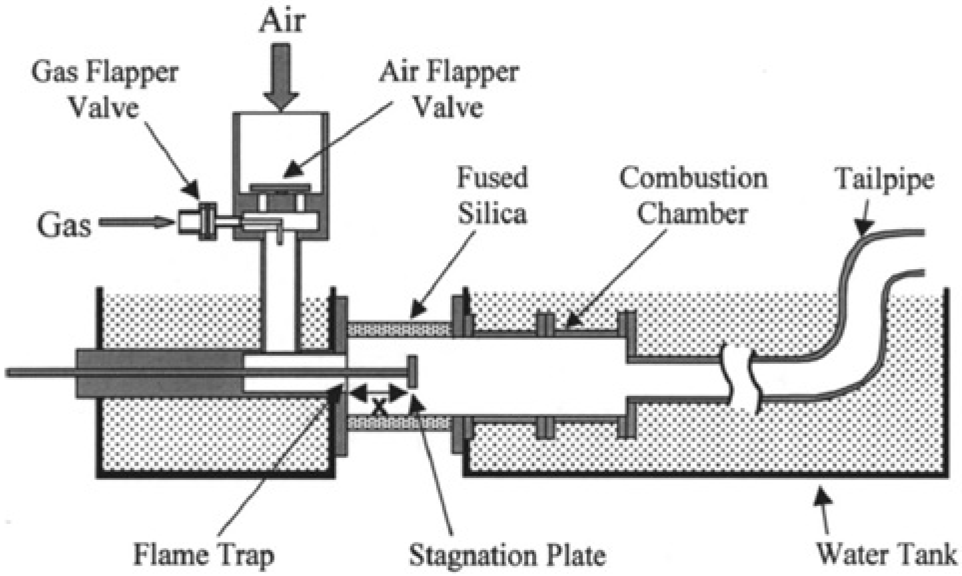

Figure 3.4: Nox removal process from pulse combustion [7]

Considerable research has been conducted into pulse combustors in a bid to understand the complex processes that govern their operation and to provide the basic understanding required to allow design strategies to be formulated. Increasingly stringent pollutant emission legislation for combustion devices is also driving research into understanding the mechanisms responsible for the inherently low NOx emissions produced by pulse combustors.

Work has shown that low NOx production is associated with the mixing fields generated during each reactant injection phase. For instance, Keller and Hongo have shown that rapid mixing between combusting gases and cooler residual gases can produce a short residence time at high temperatures, thus reducing NOx formation.

3.4 Laval Nozzle for Air flow field

Huang etal; Li andXia[14]presents, A Laval nozzle is a convergent–divergent nozzle,which can produce a supersonic flow in the divergent section, directly following the choked and sonic flow condition at the narrowest point in the nozzle. Both the scientific and patent literature have described the application of a Laval nozzle in melt blowing including various speculations regarding the production of nano- fibers. Reneker and Johnson claimed that polymer melt fibrillation within a Laval nozzle produces nanofi-bers, while Gerking and

Sodemann and Voges suggested that the spontaneous burst of a molten polymer produces multiple nanofibers. Gerking's photographic evidence of the spontaneous bursting process is inconclusive since the image could also be interpreted as fiber whipping.

Moreover, spontaneous bursting, an unexpected event for viscoelastic materials, should lead to irregularly shaped fibers, which is not found; Gerking noted that all the produced fibers have a regular cylindrical shape.

Stimulation results of LAVAL nozzle with different shapes shown in Figure 3.5

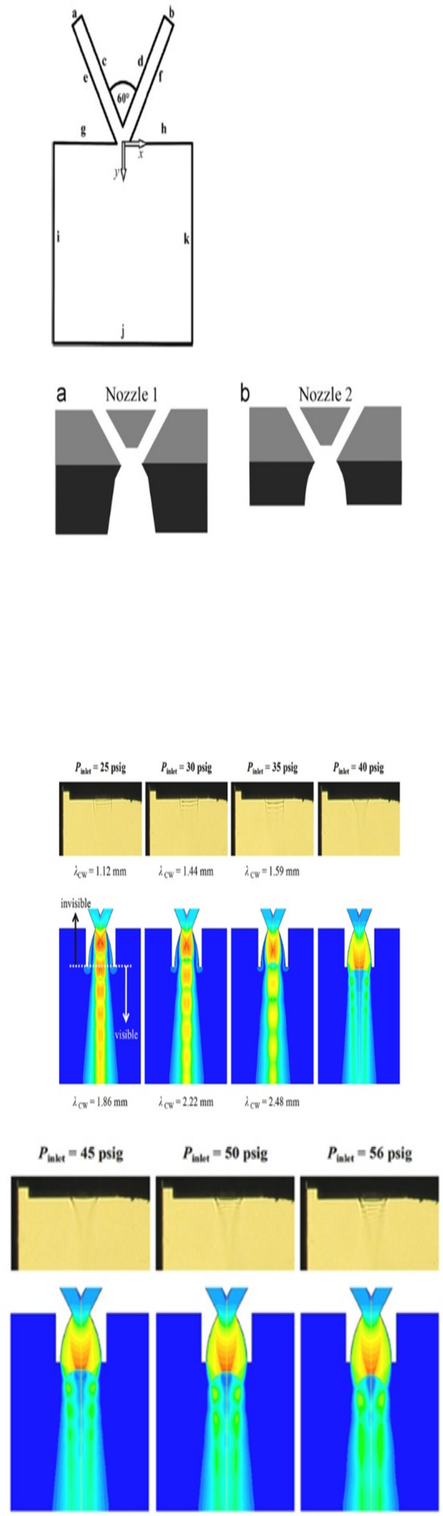

Figure 3.5: Laval nozzle with stimulation results [7]

Chapter 4

Methodology of work

4.1 Heat duty calculation:-

In the Stenter machine fabric moves from a temprature of 50°C to 180°C .When the fabric enter the chamber it is moist with water, thus

$$Total\,heat\,required\,for\,operation = Heat\,absorbed\,by\,fabric + Heat\,absorbed\,by\,moisture \tag{4.1}$$

Calculations:

1. Heat duty required for fabric drying:-

$$Mass\,flow\,rate\,of\,fabric\,(kg/s) =$$

Table 4.1: Given data

Given Data	
Area density of fabric	0.125 kg/m^2
Width of fabric	2 m
Production rate	70 m/min
Moisture contain	80% mass of fabric
Outlet temperature of hot air	220°C

Table 4.2: Heat duty calculation

Q	Q_1+ Q_2 + Q_3
Q_1	Heat duty at 50°C to 100°
Q_2	Heat duty at 100°Cto 100°C (latent heat transfer)
Q_3	Heat duty at 100°C to 180°C

$$area\,density\,of\,fabric(kg/m^2)\times production\,rate(m/min)\times width\,of\,fabric\times 1/60\,sec \tag{4.2}$$

$$Q_{fabric} = M \times C_p \times (T_{out} - T_{in}) \tag{4.3}$$

$$Q_{fabric} = 87.009\,kW$$

1. **Heat duty required for moisture removing**

- **At 50°C to 100°C**

  T_{h_i}=50°C, T_{h_o}=100°C

$$Q_{moist}(50°C\,to\,100°C) = Mass\,flow\,rate \times C_p\,of\,moisture \times (T_{out} - T_{in}) \tag{4.4}$$

$$Q_{moist}(50°C\,to\,100°C) = 0.233\,kg/sec \times 4.18\,kJ/kg.k \times (100 - 50) = 48.697\,kW$$

- **At 100°C (latent heat)**

  From steam table latent heat at 100°C =2256.9 kJ/kg

$$Q_{moist}(latent\,heat\,at\,100°C) = Mass\,flow\,rate\,of\,moisture \times (H_g - H_f) \tag{4.5}$$

$$Q_{moist} = 0.223\,kg/sec \times 2256.9\,kJ/kg = 525.85kW$$

- **At 100°C to 180°C**

$$Q_{moist}(100°C\,to\,180°C) = Mass\,flow\,rate\,of\,moisture \times C_p of\,moisture \times (T_{out} - T_{in})$$

$$Q_{moist}(100°C\,to\,180°C) = 0.233\,kg/sec \times (Enthalpy\,at\,180°C - Enthalpy\,at\,100°C)$$

$$Q_{moist}(100°C\,to\,180°C) = 0.233 \times (2835.6 - 2676.2) = 37.14\,kW$$

Finally,

$$Q_{moist} = 48.697\,kW + 525.85\,kW + 37.14\,kW = 611.687\,kW \tag{4.6}$$

$$Q_{total} = Q_{fabric} + Q_{moist} \tag{4.7}$$

$$Q_{total} = 87.009\,kW + 611.687\,kW = 698.777\,kW$$

Water will change in to the steam pressure will increase because volume will be constant but for calculation purpose we assumed pressure will be constant.

- Total heat duty required to dry the fabric = 698.777KW
- Calculation for mass flow rate of air

$$M_{air} \times C_p\,of\,air \times (T_{h_o} - T_{h_i}) = Q_{total} \tag{4.8}$$

$$M_{air} = 22.99\,kg/sec$$

20% air is exhausted so the same amount of fresh air sucked form Entry and exit of the chamber.

$$M_{fresh\,air} = 22.99 \times 0.2 = 4.6\,kg/sec$$

$$M_{dry\,air} = 22.99 - 4.6 = 18.4\,kg/sec$$

$$Total\,heat\,duty\,required\,to\,heat\,the\,air\,for\,fabric\,drying = Q_{dry\,air} + Q_{fresh\,air} \tag{4.9}$$

$$Q_{dry\,air} = M_{dry\,air} \times C_p\,of\,dry\,air \times (250 - 230) = 559.176\,kW$$

$$Q_{fresh\,air} = 4.6 \times 1.013 \times (250 - 30) = 1025\,kW$$

$$Q_{total} = 559.176 + 1025 = 1584.176\,kW$$

- Total heat duty required to heat the air for fabric drying = 1584.176KW

4.2 Blower Introduction

Fans and blowers are turbo machines which deliver air at a desired high velocity (and accordingly at a high mass flow rate) but at a relatively low static pressure. The pressure rise across a fan is extremely low and is of the order of a few millimeters of water gauge. The upper limit of pressure rise is of the order of 250mm of water gauge. The rise in static pressure across a blower is relatively higher and is more than 1000 mm of water gauge that is required to overcome the pressure losses of the gas during its flow through various passages. A blower may be constructed in multistage for still higher discharge pressure.

Typical blower for industrial use shown in Figure 4.1

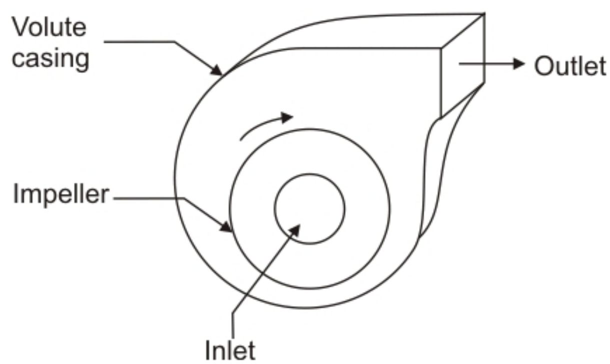

Figure 4.1: Blower used for experimental setup [11]

A large number of fans and blowers for relatively high pressure applications are of centrifugal type. The main components of a centrifugal blower are shown in Figure 4.1. A blower consists of an impeller which has blades fixed between the inner and outer diameters. The impeller can be mounted either directly on the shaft extension of the prime mover or separately on a shaft supported between two additional bearings. Air or gas enters the impeller axially through the inlet nozzle which provides slight acceleration to the air before its entry to the impeller. The action of the impeller swings the gas from a smaller to a larger radius and delivers the gas at a high pressure and velocity to the casing. The flow from the impeller blades is collected by a spiral-shaped casing known as volute casing or spiral casing . The casing can further increase the static pressure of the air and it finally delivers the air to the exit of the blower.

Blower with various equipments and direction of rotation shown in Figure 4.2

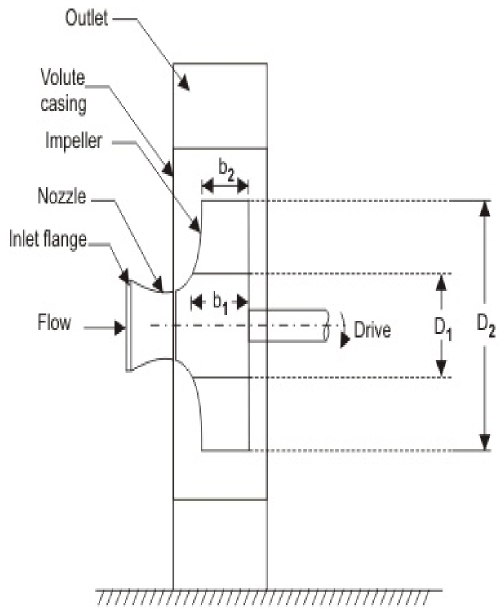

Figure 4.2: Blower side view [11]

The centrifugal fan impeller can be fabricated by welding curved or almost straight metal blades to the two side walls (shrouds) of the rotor. The casings are made of sheet metal of different thickness and steel reinforcing ribs on the outside. Suitable sealing devices are used between the shaft and the casing. A centrifugal fan impeller may have backward swept blades, radial tipped blades or forward swept blades. The inlet and outlet velocity triangles are also shown accordingly in the figure. Under ideal conditions, the directions of the relative velocity vectors and are same as the blade angles at the entry and the exit.

A zero whirl at the inlet is assumed which results in a zero angular momentum at the inlet. The backward swept blades are employed for lower pressure and lower flow rates. The radial tipped blades are employed for handling dust-laden air or gas because they are less prone to blockage, dust erosion and failure. The radial-tipped blades in practice are of forward swept type at the inlet. The forward-swept blades are widely used in practice. On account of the forward-swept blade tips at the exit, the whirl component of exit velocity is large which results in a higher stage pressure rise.

The following observations may be noted

$$V_{w2}<U_2, \text{ if } ,\beta_2<90^0 \text{backward swept blades },$$

$$V_{w2}=U_2, \text{ if } ,\beta_2=90^0 \text{if , radial blades },$$

$V_{w2} > U_2$, if , $\beta_2 > 90^0$ if , forward swept blades ,

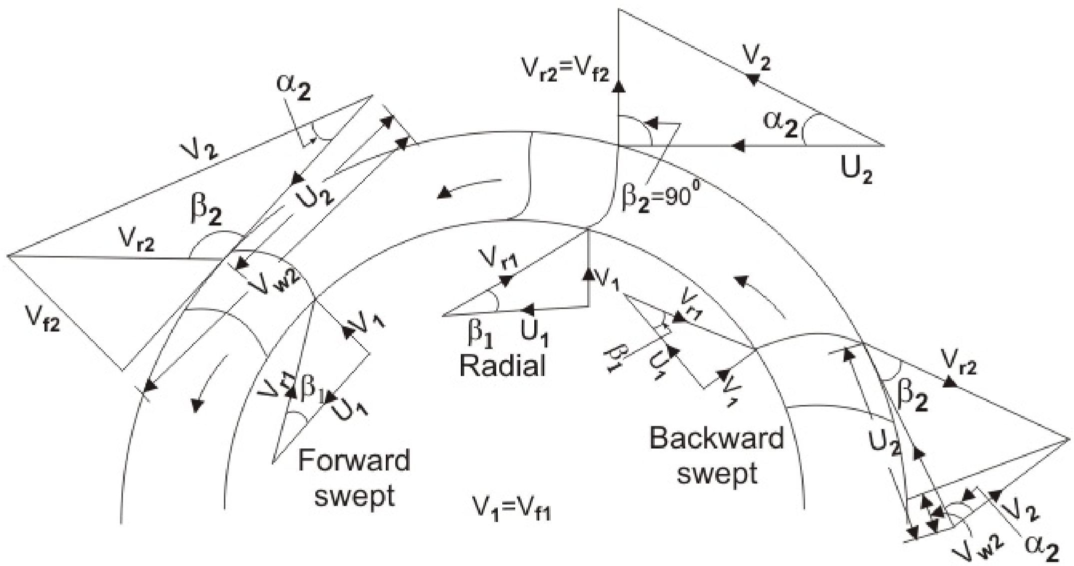

Figure 4.3: Backward,Radial,Forward blades Arrangement [11]

The mass flow rate through the impeller is given by

$$\dot{m} = \rho_1 Q_1 = \rho_2 Q_2 \tag{4.10}$$

The areas of cross sections normal to the radial velocity components V_{f1} and V_{f1} are $A_1 = \pi D_1 b_1$ and $A_2 = \pi D_2 b_2$

$$m = \rho_1 V_{f2} (\pi D_2 b_2) \tag{4.11}$$

The radial component of velocities at the impeller entry and exit depend on its width at these sections. For small pressure rise through the impeller stage, the density change in the flow is negligible and the flow can be assumed to be almost incompressible. For constant radial velocity

$V_{f1} = V_{f1} = V_f$

4.2.1 Work

The work done is given by Euler's Equation

$$w = U_2V_{w2} - U_1V_{w1} \tag{4.12}$$

It is reasonable to assume zero whirl at the entry. This condition gives

$\alpha 1 = 90^0$,V_{w1}=0 And hence $U_1V_{w1} = 0$

Therefore we can write,

$$V_1 = V_{f1} = V_{f2} = U_1 \tan\beta_1 \tag{4.13}$$

Equation gives

$$w = U_2V_{w2} = U_2^2(\frac{V_{w2}}{U_2}) \tag{4.14}$$

For any of the exit velocity triangles

$$U_2 - V_{w2} = V_{f2}\cot\beta_2 \tag{4.15}$$

$\frac{V_{w2}}{U2}$=[1-$\frac{V_{f2}\cot\beta_2}{U_2}$]

$$w = U_2^2[1 - \phi\cot\beta_2] \tag{4.16}$$

Where$\phi = (\frac{V_{f2}}{U_2})$ is known as flow coefficient

Head developed in meters of air $H_a = \frac{V_wU_2}{g}$

Equivalent head in meters of water $H_w = \frac{\rho_a H_a}{\rho_w}$

Where $\rho_a and\ \rho_w$ are the densities of air and water respectively.

Assuming that the flow fully obeys the geometry of the impeller blades, the specific work done in an isentropic process is given by

$$\triangle h_0 = U_2(1 - \cot\beta_2) \tag{4.17}$$

Power required to drive fan is

$$P = m(\triangle h_0) = mU_2V_{w2} = mU_2^2(1 - \phi\cot\beta_2) = mC_p(\triangle T_0) \tag{4.18}$$

The static pressure rise through the impeller is due to the change in centrifugal energy and the diffusion of relative velocity component. Therefore, it can be written as

$$P_2 - P_1 = (\triangle p) = \frac{1}{2}\rho(U_2^2 - U_1^2) + \frac{1}{2}(V_{r1}^2 - V_{r2}^2) \tag{4.19}$$

$$(\triangle p_0) = \frac{1}{2}\rho(U_2^2 - U_1^2) + \frac{1}{2}(V_{r1}^2 - V_{r2}^2) + \frac{1}{2}\rho(V_2^2 - V_1^2) \tag{4.20}$$

The stagnation pressure rise through the stage can also be obtained as

$$(\triangle p_0) = (\triangle p) + \frac{1}{2}\rho(V_1^2 - V_2^2) \tag{4.21}$$

From any of the outlet velocity triangles

$$\frac{V_2}{\sin\beta_2} = \frac{U_2}{\sin\{\pi-(\alpha_2+\beta_2)\}}$$

$$\frac{V_2}{\sin\beta_2} = \frac{U_2}{\sin(\alpha_2+\beta_2)}$$

$$\mathrm{V}_{w2} = V_2\cos\alpha_2 = \frac{U_2\sin\beta_2\cos\alpha_2}{\sin(\alpha_2+\beta_2)}$$

$$\frac{V_{w2}}{U_2} = \frac{\sin\beta_2\cos\alpha_2}{\sin\alpha_2\cos\beta_2+\cos\alpha_2\sin\beta_2}$$

$$\frac{V_{w2}}{U_2} = \frac{\tan\beta_2}{\tan\alpha_2+\tan\beta_2}$$

$$w = U_2^2(\frac{\tan\beta_2}{\tan\alpha_2 + \tan\beta_2}) \tag{4.22}$$

4.2.2 Efficiency

On account of losses, the isentropic work is less than the actual work $\frac{1}{\rho}(\triangle p_0)$ is less than actual work

Therefore the stage efficiency is defined by

$$\eta_s = \frac{(\triangle p_0)}{\rho U_2 V_{w2}} \tag{4.23}$$

4.2.3 Number of Blades

Too few blades are unable to fully impose their geometry on the flow, whereas too many of them restrict the flow passage and lead to higher losses. Most of the efforts to determine the optimum number of blades have resulted in only empirical relations [1] given below

$$n = \frac{8.5 \sin \beta_2}{1 - \frac{D_1}{D_2}} \tag{4.24}$$

$$n = 6.5(\frac{D_2 + D_1}{D_2 - D_1}) \sin \frac{1}{2}(\beta_1 + \beta_2) \tag{4.25}$$

4.2.4 Impeller Size

The diameter ratio (D_1/D_2)of the impeller determines the length of the blade passages. The smaller the ratio the longer is the blade passage. The following value for the diameter ratio is often used by the designers

$$\frac{D_1}{D_2} = 1.2(\phi)^{\frac{1}{3}} \tag{4.26}$$

where

$$\phi = V_{f2}/U_2$$

The following relation for the blade width to diameter ratio is recommended:

$$\frac{b_1}{D_1} = .2$$

If the rate of diffusion in a parallel wall impeller is too high, the tapered shape towards the outer periphery, is preferable

4.2.5 Results

From design calculation various dimension of blower were worked out which is summarized in Table 4.3

Table 4.3: Blower specification

m (kg/sec)	4.7	D1 (m)	.80544	ψ	.19141
q (m^3/hr)	22000	B1 (m)	.161089	U_2(m/sec)	77.25
C_1(m/sec)	15	N (Rpm)	509.1566	$\triangle p_2$	3248.04
$\beta_1(radian)$	.61	n	9	B2 (m)	.1611089
C_{r1}(m/sec)	15	M	2.88	C_2(radian)	56
C_{r2}(m/sec)	15	D(m)	23.25	B	.5229
ρ (kg/m^3)	.779	$C_{\theta 2}$	54.5	w	4187.19
U_1(m/sec)	21.46	α_2(radian)	.2678	ψ_{st}	1.3
w_1(m/sec)	26.184	P (Kw)	19.86	R	.6509
β_2(radian)	.572826	D2 (m)	23.26	η_{st}	.993
U_2(m/sec)	77.25	$\triangle p_1$	2114.212		

Chapter 5

CFD Analysis of Fan House of Stenter Machine

5.1 Introduction

CFD is one of the branch of fluid mechanics that uses numerical methods and algorithms to solve and analyze problem that involve fluid flow. The CFD approach in one or another form, is based on fundamental governing equations of fluid dynamics as mentioned below:

- Continuity Equation

- Momentum and Energy equations.

1. Mass conservation equation

States that for any system closed to all transfers of matter and energy (both of which have mass), the mass of the system must remain constant over time, as system mass cannot change quantity if it is not added or removed. Hence, the quantity of mass is "conserved" over time. The law implies that mass can neither be created nor destroyed, although it may be rearranged in space, or the entities associated with it may be changed in form, as for example when light or physical work is transformed into particles that contribute the same mass to the system as the light or work had contributed. The law implies (requires) that during any chemical reaction, nuclear reaction, or radioactive decay in an isolated system, the mass of the reactants or starting material must equal the mass of the products.

$$Continuity \quad \frac{\partial \rho}{\partial t} + div(\rho v) = 0 \tag{5.1}$$

2. Momentum conservation equation

Conservation of momentum is a fundamental law of physics which states that the momentum of a system is constant if there are no external forces acting on the system.

$$X - momentum \quad \frac{\partial(\rho u)}{\partial t} + div(\rho u u) = -\frac{\partial p}{\partial x} + div(\mu grad u) + S_{Mx} \tag{5.2}$$

$$Y - momentum \quad \frac{\partial(\rho v)}{\partial t} + div(\rho v u) = -\frac{\partial p}{\partial y} + div(\mu grad v) + S_{My} \tag{5.3}$$

$$Z - momentum \quad \frac{\partial(\rho w)}{\partial t} + div(\rho w u) = -\frac{\partial p}{\partial z} + div(\mu grad w) + S_{Mz} \tag{5.4}$$

3. Energy conservation equation

It states that the total amount of energy in an isolated system remains constant over time. The total energy is said to be conserved over time. For an isolated system, this law means that energy is localized and can change its location within the system, and that it can change form within the system, for instance chemical energy can become kinetic energy, but that it can be neither created nor destroyed. Moreover, two initially isolated systems, that have no external or mutual interaction, can be logically composed into a single isolated system, and then the total amount of energy of the composite system is equal to the sum of the respective total amounts of energy of the two component systems. In this sense, the energy of a system is said to be additive.

The set of equations which describe the processes of momentum, heat and mass transfer are known as Navier Stroke's equations. The most common in this technique, is known as the finite volume technique. The region of interest is divided into small sub regions called control volumes. Now the equations are discretized and solved iteratively for each control volumes. So finally approximation of the value of each variable at specific point throughout the domain can be obtained.

$$[u\frac{\partial T}{\partial x} + v\frac{\partial T}{\partial y} + w\frac{\partial T}{\partial z}] = \frac{1}{\alpha}[\frac{\partial^2 T}{\partial x^2} + \frac{\partial^2 T}{\partial y^2} + \frac{\partial^2 T}{\partial z^2}] \tag{5.5}$$

5.2 Structure of CFD code

CFD codes are structured around the numerical algorithms that can tackle fluid flow problems. All commercial CFD packages include sophisticated user interfaces to input problem parameters and to examine the results. Hence all codes contain three main elements: (i) a pre-processor, (ii) a solver and (iii) a post-processor. The function of each of these elements is briefly described, within the context of a CFD code.

Pre-processor :

Pre-processing consists of the input of a flow problem to a CFD program by means of an operator-friendly interface and the subsequent transformation of this input into a form suitable for use by the solver. The user activities at the pre-processing stage involve:

- Definition of the geometry of the region of interest : the computational domain
- Grid generation - the sub-division of the domain into a number of smaller, non-overlapping sub-domains
- Selection of the physical and chemical phenomena that need to be modeled.
- Definition of fluid properties
- Specification of appropriate boundary conditions at cells which coincide with or touch the domain boundary

Solver:

There are three distinct streams of numerical solution techniques: finite difference, finite element and spectral methods. In outline the numerical methods that form the basis of the solver perform the following steps:

- Integration of the governing equations of fluid flow over all the control volumes of the domain
- Discretisation - conversion of the resulting integral equations into a system of algebraic equations
- Solution of the algebraic equations by an iterative method

Post-processor:

As in pre-processing a huge amount of development work has recently taken place in the post-processing field. flowing to the increased popularity of engineering work stations, many of which have outstanding graphics capabilities, the leading CFD packages are flow equipped with versatile data visualization tools. These include:

- Domain geometry and grid display
- Vector plots
- Line and shaded contour plots
- 2D and 3D surface plots
- Particle tracking
- View manipulation (translation, rotation, scaling etc.)
- Colour postscript output

5.3 Typical Steps of CFD Stimulation in Software

The typical steps of solving the problem using CFD software are under:

1. Create the geometry model and mesh it.

2. Start the appropriate solver for 2D or 3D modeling.

3. Import the grid and check it.

4. Select the solver formulation

5. Chose the basic equation to solve: laminar or turbulent (or in viscid), chemical species or reaction, heat transfer models, etc. Also identify additional models needed: fans, heat exchangers, porous media, etc.

6. Specify the material properties.

7. Specify the boundary properties.

8. Adjust the solution control parameter.

9. Initialize the flow field.

10. Calculate a solution.

11. Examine the results.

12. Save the results.

13. If necessary, refine the grid or consider revisions to the numerical or physical

Some simplifying assumptions are required before applying the conventional Navier-Stokes and Energy equations to the model. The major assumptions are:

(1) Steady state flow and heat transfer,

(2) Incompressible fluid,

(3) Laminar flow,

(4) Uniform wall heat flux,

(5) Constant solid and fluid properties (thermo physical properties)

5.4 CFD Analysis of Fan House of Stenter Machine

The Stenter machine usually consists of 8-10 chamber and each chamber contains 2-blowers, each blower provided in seperate casing and each casing contains 12 nozzles, i.e total 24 nozzles are provided in each chamber (12 facing down and 12 facing up). In each nozzle there is 48 openings, each of which are supplying the air to the fabric. The view of Stenter machine shown in Figure 5.1

In the present, the CFD analysis carried out in 3 stages as mentioned below;

Stage 1: Fan B with casing

Stage 2: Fan A with casing

Stage 3: Fan A with Nozzle

Figure 5.1 showing section stenter used by IEPL, in this Figure 5.1 4 chamber are there with 8 blowers

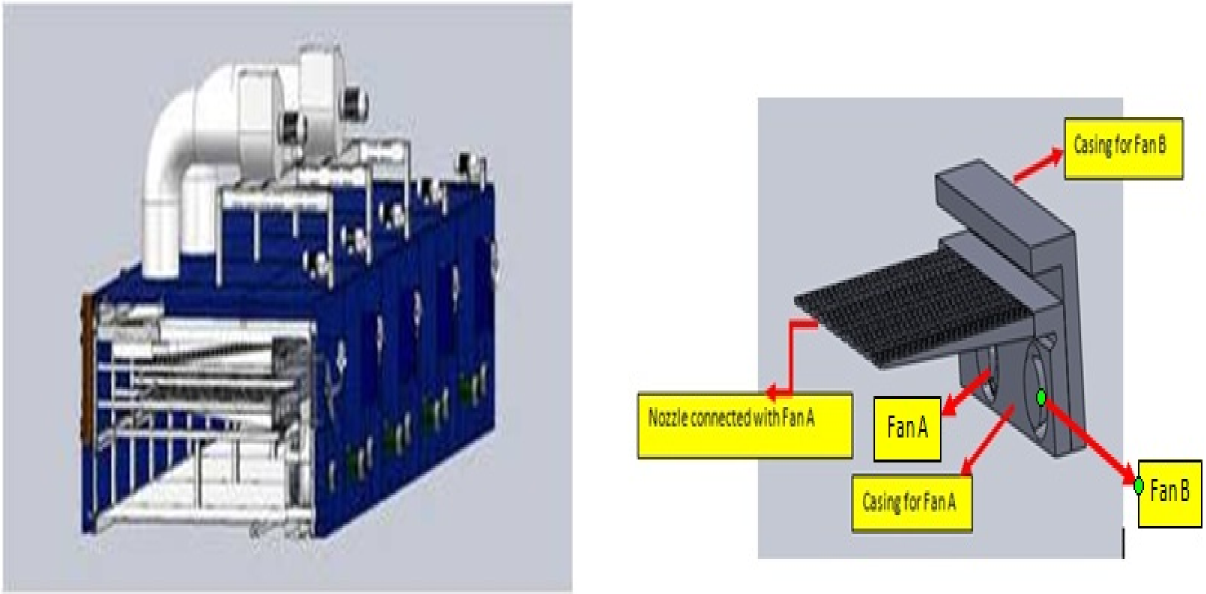

Figure 5.1: Section view of stenter and aarangement of Fan A and Fan B

5.4.1 Stage 1: Fan B With Casing

5.4.1.1 Geometry Creation

Geometry of Fan B with casing is generated in Solidworks software. Steps are followed as given below:

Step 1 : 3-D drawing of casing and fan is generated with help data provided by InspirOn Engg Pvt Ltd.

Step 2 : Rectangle of Dimension 900x 2880 is created, after this extrude command is adopted to generate solid.

Step 3: Two circle of dimension 630mm cutted,by using extrude cut command.

Step 4: Impeller part is created,using different command method.

Step 5: Another circle is created right side of casing of diameter 632mm ,so that impeller can be assembled.

Step 6: Whole body is united by unite command to make a single volume,there would be only 2 volumeone is casing another is impeller.

Step 7: Geometry is exported in .STEP to Gambit 2.4 for meshing

Computational domain after mesh generation shown in Figure 5.2

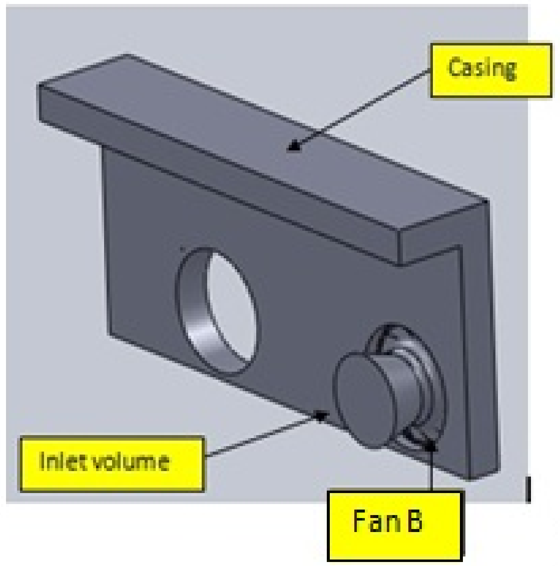

Figure 5.2: Computatinal domain for FAN B with casing

Proper assembly of casing and impeller is very important in case of CFD analysis. Inner face of volute casing and outer face of impeller should be matched. Any discrepancy in the assembly can lead to unexpected result. Assembly is done by matching the point on inner face of casing to the corresponding point on outer face of impeller.

for assemblying nozzle with casing,nozzle assembly is created and mate command is adopted to attach nozzle assembly with casing

5.4.1.2 Grid Generation

One of the most important and time-consuming task in the CFD simulation process is the generation of the computational grid or mesh. As the geometry is complex unstructured grid consists of triangular and tetrahedral element is used. Grid is generated in Altair Hypermesh 9.0 software.

The dynamic mesh model in FLUENT can be used to model flows where the shape of the domain is changing with time due to motion on the domain boundaries. The motion can be a prescribed motion (e.g., you can specify the linear and angular velocities about the center of gravity of a solid body with time) or an unprescribed motion where the subsequent motion is determined based on the solution at the current time (e.g., the linear and angular velocities are calculated from the force balance on a solid body).

Grid of Fan B generated in Hyperworks software shown in Figure 5.3

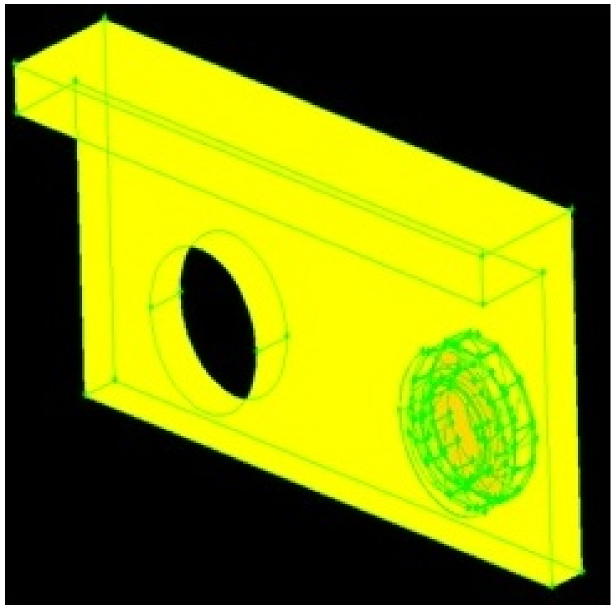

Figure 5.3: Grid generation of FAN B with casing

The update of the volume mesh is handled automatically by FLUENT at each time step based on the new positions of the boundaries. To use the dynamic mesh model, you need to provide a starting volume mesh and the description of the motion of any moving zones in the model. FLUENT allows to describe the motion using either boundary problems or user-defined functions (UDFs). FLUENT expects the description of the motion to be specified on either face or cell zones.

If the model contains moving and non-moving regions, need to identify these regions by grouping them into their respective face or cell zones in the starting volume mesh that you generate.

Furthermore, regions that are deforming due to motion on their adjacent regions must also be grouped into separate zones in the starting volume mesh. The boundary between the various regions need not be conformal. Can use the non conformal or sliding interface capability in FLUENT to connect the various zones in the final model.

Various meshing size and type is adopted while doing CFD of Fan B,Appropiate meshing adopted and discussed in Table 5.1.

Table 5.1: Details of grid

Name of component	**Type of grid**	**Grid size**	**No. of elements**
FAN A			
Casing	Tetrahedral	2mm	25,00,000
Impeller	Tetrahedral	2mm	13,00,000
Total			38,00,000

5.4.1.3 Zone Specification

After meshing geometry is imported in Gambit for boundary condtions. Zone-type specifications define the physical and operational characteristics of the model at its boundaries and within specific regions of its domain. There are two classes of zone-type specifications:

- Boundary types
- At inlet of impeller: Mass flow inlet
  - At outlet of casing : Static pressure outlet
  - Casing walls
  - Impeller walls
  - At the exit of Impeller: Wall 1, which will be changed to Interface in FLUENT
  - At the inlet to Casing: Wall 2, which will be changed to Interface in FLUENT
- Continuum types
  - Runner volume : Fluid zone 1
  - Casing : Fluid zone 2

Boundry condition for Fan B shown in Figure 5.4

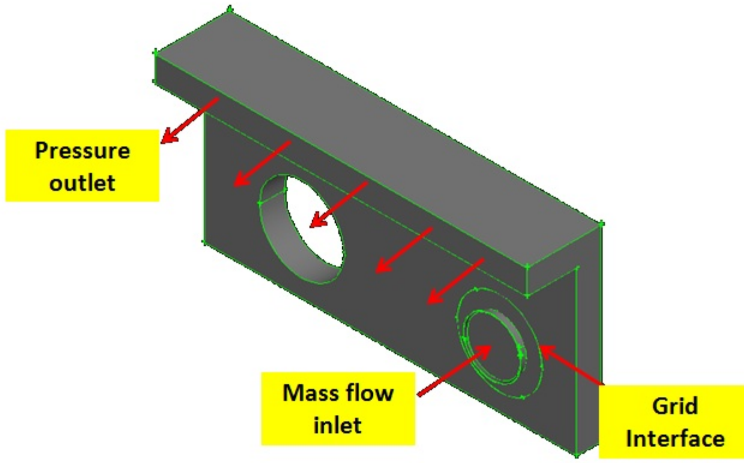

Figure 5.4: Zone specification of Fan B

After completion of modeling, meshing and specifying boundary zones in GAMBIT, the mesh is exported to 'FLUENT 5/6 Solver' for simulation.

5.4.1.4 Simulation in Fluent

With the aid of CFD, the complex flow in the different components of pump can be well predicted. Thus it is very useful in the design of pumps. This article describes the 3-dimensional simulation of ow in Centrifugal Pump. A commercial 3-dimensional Navier-Stokes code called 'FLUENT' with different turbulence models is used to simulate the flow. The FLUENT's Moving Reference Frame (MRF) - model is used to consider the Runner in rotating reference frame and other components in stationary reference frame. In calculation, Finite Volume method (FVM) is used for the discretization of governing equations.

5.4.1.5 Assumptions

The following assumptions were made during the simulation of Centrifugal Pump.

- Flow is Steady state.
- Fluid is incompressible.
- Fluid properties are constant.
- No vapor is present in the water, i.e. single phase flow.

- There is no leakage in the whole pump.
- The surface of all the components is hydraulically smooth.

5.4.1.6 Solution Technique

Following technique is used for the simulation.

Considering the available computational facilities, FLUENT's single precision version is used. As such, the double precision version is more accurate, because it is free from round-off errors, but it requires more memory.

- The grid is read and checked in FLUENT.
- The segregated implicit solver with absolute velocity formulation is used for the computation. The node-based gradient scheme is used which is more accurate than the cell-based scheme for unstructured meshes, most notably for triangular and tetrahedral meshes.
- Various inputs in FLUENT software are listed below:.
  - Mass flow inlet 2.4kg/sec
  - Pressure outlet 1000 pascal
- Impeller is taken in moving reference frame, rotating at the speed of 1500 rpm.
- Direction -Clockwise
- Water with following properties is used as a working fluid.
  - Density .7785 kg/m3
  - Viscosity 0.002551 kg/m-s
- The casing wall & impeller wall were taken in stationary reference frame.
- At the junction between two fluid zones, non-conformal mesh is used. Hence interfaces are defined, between runner inlet and casing inlet
- Following convergence criteria is used for different equations.
  - Continuity equation 0.001

- Momentum equations 0.001
- Turbulence kinetic energy 0.001
- Turbulence dissipation rate 0.001

- A SIMPLE scheme is used for the pressure velocity coupling.
- For momentum and turbulence, second order upwind scheme is used.
- Following under relaxation factors are used.
  - Pressure 0.3
  - Momentum 0.5
  - Turbulence kinetic energy 0.5
  - Turbulence dissipation rate 0.5
- Operating conditions 101325 Pa pressure is used as operating condition outside the computational domain.

5.4.2 Results and Discussion

5.4.2.1 Computational Parameters

Flow rate is varied with the help of flow regulating valve. At different flow rates readings of pressure and energy meter readings are noted. Power input is calculated by equation.

$$Power\,input = \frac{2 \times \pi \times N \times T}{60} \tag{5.6}$$

Output power is calculated by equation,

$$Power\,output = \rho \times g \times Q \times H \tag{5.7}$$

Efficiency ,

$$Efficiency = \frac{Power\,output}{Power\,input} \tag{5.8}$$

5.4.2.2 Grid Independency Test [16]

Grid independency test is done to check the effect of grid size on the numerical result. Here CFD analysis of fan housing of stenter at design condition with standard k-ϵmodel is done with different size of grid and compared the results with the model testing results. If the results between two finest meshes are nearly equal then it is considered to be grid independent.

The model is tested for grid-independence to give proper resolution to the region where large gradients of fluid flow r characteristic is predicted. The optimum grid system has the meshing resolution. The model in this study uses a total number of grid 37, 00000. A grid independence test was carried out by doubling the grid size for the Stenter with the dimension discussed earlier.

The fine grid mesh for the x , y-directions and z-direction is adopted to properly resolve the velocity .The meshing along the surfaceof entrance region a relatively T- grid system has taken.

The reasons for the T-grid discretization for the x,y, z-direction are, with the exception of the inlet region, are small compared to the gradients occurring in other directions and the CPU time as well as the memory storage required increases dramatically as the number of grid nodes is increased.

Furthermore comparison with standard numerical results, Indicates that the finer the mesh size, the higher the numerical accuracy.

To go ahead in CFD process grid independency is conducted, the summary of which is given in Table 5.13

Table 5.2: Grid independency test

Grid	No. of elements	Total Head (mm)	Power (kW)	Efficiency (%)
Model testing		102	4.53	67.56
First type	17,12,572	73	4.0	41.2
Second type	25,82,952	73.5	4.2	47.6
Third type	**27,26,982**	**75**	**4.6**	**50**
Fourth type	35,21,845	NA	NA	NA

It can be seen from above table that, results obtained with third type grid were closer to model testing results. Also as number of grid elements increses some change in the results was observed, however due to computatinal resources limitation final grid (4th type) could not be used.

5.4.2.3 Variation in Pressure and Velocity

The fluid is entered through the inlet at mass flow rate 2.4 kg/sec with with outlet pressure 1000 pascal. After passing through the casing, the fluid discharged to the atmosphere i.e gauge pressure.

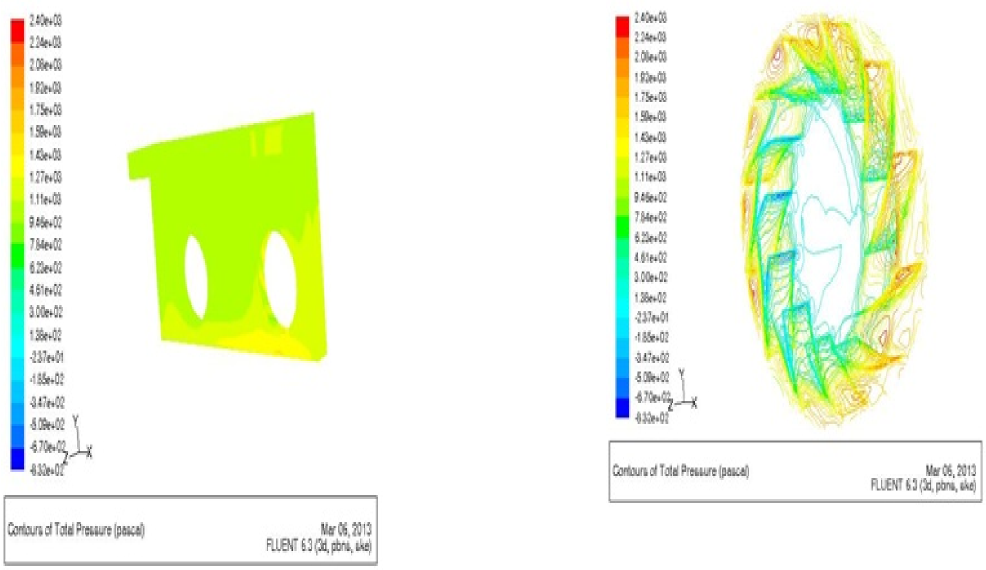

Figure 5.5: Pressure contours of casing (left) and impeller (right) of Fan B

Figure 5.5 shows pressure contours in casing and impeller of Fan B respectively. It can be seen that as air moves radially outward in the impelller pressure increases. In casing non-uniform distribution of pressure was observed.

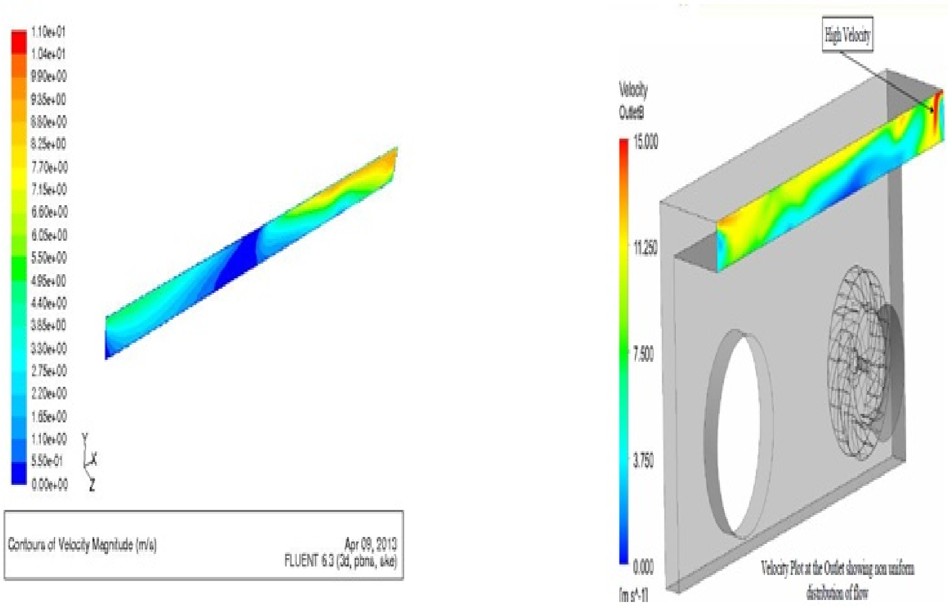

Figure 5.6: Velocity contours of present study (left) and provided by IEPL (left)

Figure 5.6 shows comparison of results obtained for velocity contours at casing outlet, in present study with results provided by IEPL. It can be seen that non-uniform distribution of air at casing outlet exits, similar results were obtained by IEPL which shows quiet good agreement of results.

5.4.3 Stage 2: Fan A with Casing

5.4.3.1 Geometry Creation

The geometry of Fan A with casing was created by following the steps similar to that followed for analysis of Fan B with (stage-1)

Geometry Of Fan A is created in solidwork software shown in Figure 5.7

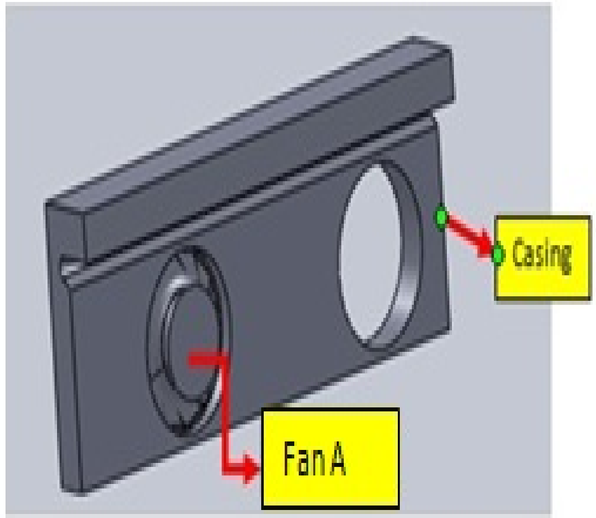

Figure 5.7: Stepwise procedure creating geometry of Fan A with casing

Meshing of FAN A with Tetrahedal grid is shown in Figure 5.8

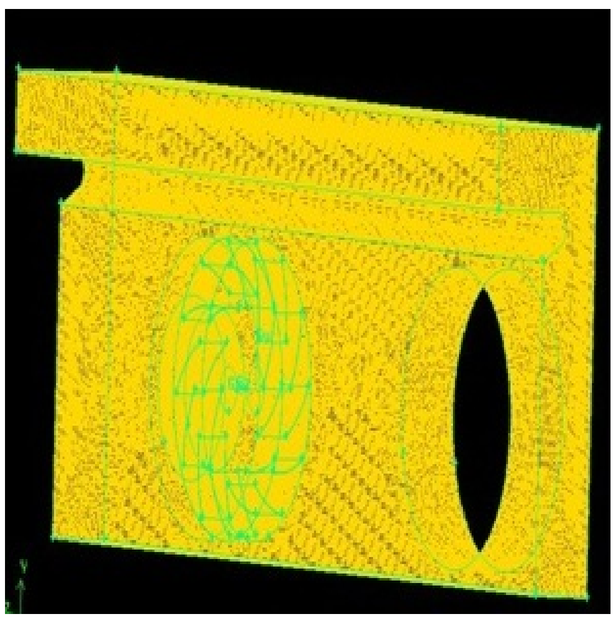

Figure 5.8: Meshing of FAN A with casing

The details of grid are given in Table 5.3

Table 5.3: Details of grid

Name of component	Type of grid	Grid size	No. of elements
FAN A			
Casing	Tetrahedral	2mm	17,00,000
Impeller	Tetrahedral	2mm	11,00,000
Total			28,00,000

5.4.3.2 Zone Specification

After meshing geometry is imported in Gambit for boundary condtions. Zone-type specifications define the physical and operational characteristics of the model at its boundaries and within specific regions of its domain. There are two classes of zone-type specifications:

Boundry condition Fan A , inlet, outlet, and interface are shown in Figure 5.9

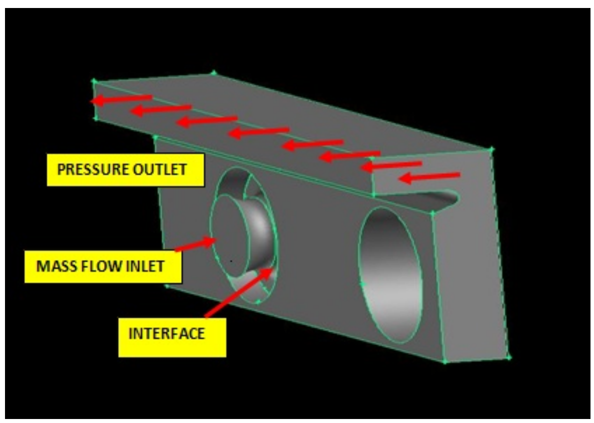

Figure 5.9: Boundry condition of Fan A

- Boundary types
- At inlet of impeller: Mass flow inlet
  - At outlet of casing : Static pressure outlet
  - Casing walls
  - Impeller walls
  - At the exit of Impeller: Wall 1, which will be changed to Interface in FLUENT
  - At the inlet to Casing: Wall 2, which will be changed to Interface in FLUENT

- Continuum types
  - Runner volume : Fluid zone 1
  - Casing : Fluid zone 2

After completion of modeling, meshing and specifying boundary zones in GAMBIT, the mesh is exported to 'FLUENT 5/6 Solver' for simulation.

5.4.3.3 Assumption, Boundry Condition and Solution Technique

Assumption, boundry condition and except direction of rotation of impeller in solution technique, were similar to analysis of stage-1

5.4.3.4 CFD Results

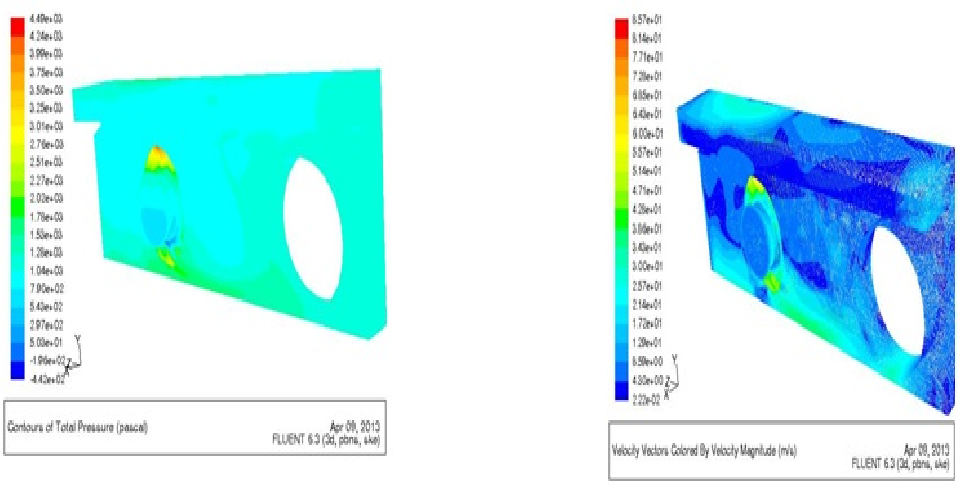

Figure 5.10: Pressure and velocity contours of Fan A with casing

Figure 5.10 shows pressure contours in casing and impeller of Fan A respectively. It can be seen that as air moves radially outward in the impelller pressure increases. In casing non-uniform distribution of pressure was observed.

Comparison of outlet wall with different colours is shown Figure5.11

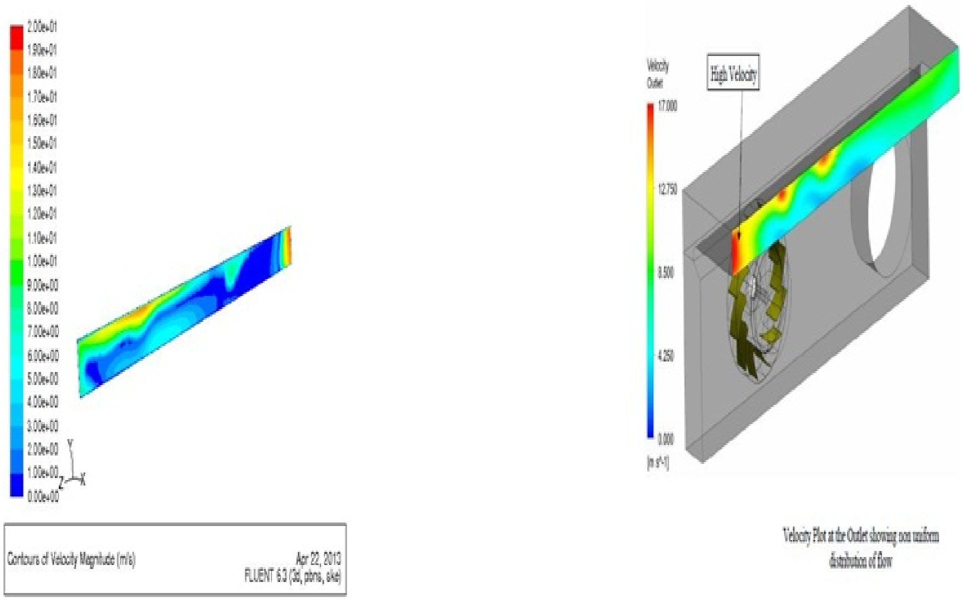

Figure 5.11: Result validation,outlet wall with IEPL,Stage 2

Figure 5.11 shows comparison of results obatined for velocity contours at casing outlet, in present study with results provided by IEPL it can be seen that non-uniform distribution of air at casing outlet exits, similar results were obtained by IEPL which shows quiet good agreement of results.

5.4.4 Stage-3: Fan A with Nozzle

5.4.4.1 Geometry Creation

The geometry of Fan A with 12 nozzles was created in Solidworks software as shown in Figure 5.12

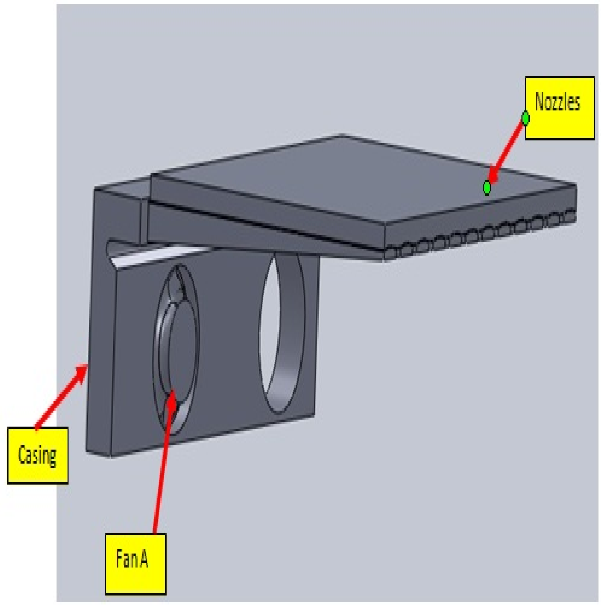

Figure 5.12: Fan B with Nozzle

Proper assembly of casing ,impeller with nozzle is very important in case of CFD analysis. Inner face of volute casing and outer face of impeller should be matched and face of nozzle volume should unite with casing,impeller assembly. Any discrepancy in the assembly can lead to unexpected result. Assembly is done by matching the point on inner face of casing to the corresponding point on outer face of impeller.

5.4.4.2 Grid Generation

The dynamic mesh model in FLUENT can be used to model flows where the shape of the domain is changing with time due to motion on the domain boundaries. The motion can be a prescribed motion (e.g., you can specify the linear and angular velocities about the center of gravity of a solid body with time) or an unprescribed motion where the subsequent motion is determined based on the solution at the current time (e.g., the linear and angular velocities are calculated from the force balance on a solid body).

Meshing of FAN A with nozzle with Tetrahedal grid with front view,top view,side view is shown Figure 5.13

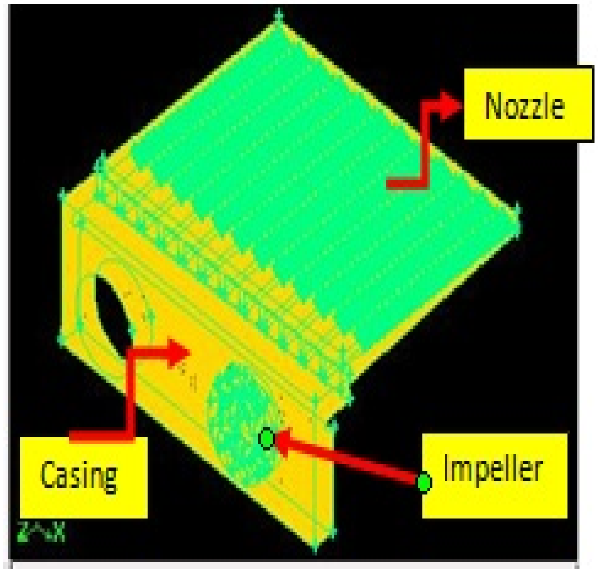

Figure 5.13: Grid generattion of Fan A with Nozzle

The update of the volume mesh is handled automatically by FLUENT at each time step based on the new positions of the boundaries. To use the dynamic mesh model, you need to provide a starting volume mesh and the description of the motion of any moving zones in the model. FLUENT allows to describe the motion using either boundary problems or user-defined functions (UDFs). FLUENT expects the description of the motion to be specified on either face or cell zones.

If the model contains moving and non-moving regions, need to identify these regions by grouping them into their respective face or cell zones in the starting volume mesh that you generate.

Furthermore, regions that are deforming due to motion on their adjacent regions must also be grouped into separate zones in the starting volume mesh. The boundary between the various regions need not be conformal. Can use the non conformal or sliding interface capability in FLUENT to connect the various zones in the final model.

Different components of fan B with casing are meshed brief discrption of components and type,grid size,number of elements are given in Table 5.4

Table 5.4: Details of grid

Name of component	Type of grid	Grid size	No. of elements
FAN B WITH CASING			
Casing	Tetrahedral	2mm	33,00,000
Impeller	Tetrahedral	2mm	3,00,000
Inlet Volume	Tetrahedral	2mm	1,00,000
Total			37,00,000

5.4.4.3 Zone Specification

Same type of zone are adopted as discussed in stage-1 and stage-2.

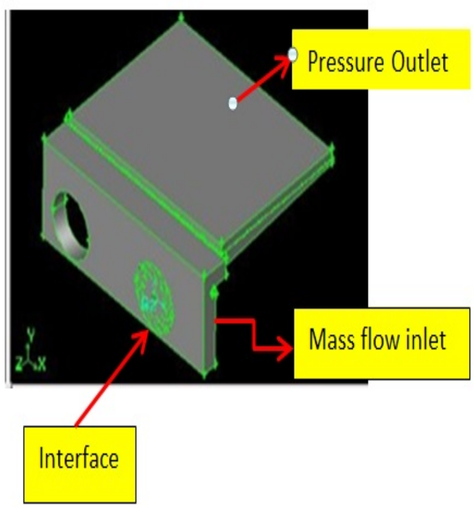

Figure 5.14: Boundry condition of Fan A

5.4.4.4 Export the Mesh

After completion of modeling, meshing and specifying boundary zones in GAMBIT, the mesh is exported to 'FLUENT 5/6 Solver' for simulation.

5.4.4.5 Simulation in Fluent

With the aid of CFD, the complex flow in the different components of pump can be well predicted. Thus it is very useful in the design of pumps. This article describes the 3-

dimensional simulation of ow in Centrifugal Pump. A commercial 3-dimensional Navier-Stokes code called 'FLUENT' with different turbulence models is used to simulate the flow. The FLUENT's Moving Reference Frame (MRF) - model is used to consider the Runner in rotating reference frame and other components in stationary reference frame. In calculation, Finite Volume method (FVM) is used for the discretization of governing equations.

5.4.5 Assumptions

The assumption made were similar to that of stage-1 and stage-2 analysis.

5.4.6 Solution Technique

The solution technique made were similar to that of stage-1 and stage-2 analysis.

5.4.7 CFD Results

Velocity vectors and pressure contours of Fan A described briefly in Figure 5.15

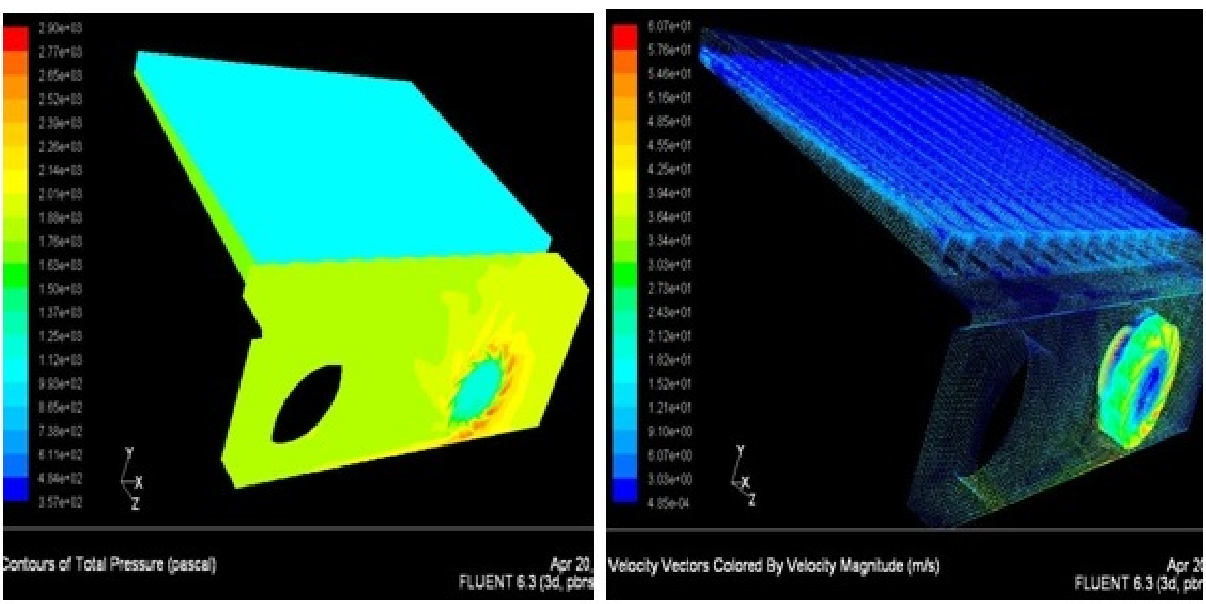

Figure 5.15: Pressure and velocity contours of Fan A with nozzle

5.4.8 Comparison of Results

The comparison of results with the experimental and CFD results provided by IEPL is shown in Table 5.5

Table 5.5: Comparison of CFD results with IEPL

	IEPL		**Present Study (CFD analysis)**	
	Experimental Results	CFD Results	Only nozzle	Nozzle + Casing
Nozzle Inlet velocity (m/sec)	7.1 m/s	-	6.83017 m/s	10 m/s
Nozzle Outlet velocity (m/sec)	24 to 28 m/s	-	25 to 28 m/s	38 to 40 m/s
Velocity at fabric (m/sec)	-	-	8.5 to 13 m/s	3 to 13 m/s
Power Input (kW)	4.53	3.297	4.2	
Torque (Nm)	-	21	27.4	
Pressure Head (mmWC)	102	52.5	70.5	
Efficiency (%)	67.85	49.48	50	

Chapter 6

Conclusions and Future Work

6.1 Conclusions

The Stenter machine is used for the drying of woven and knitted fabrics. Four type of process are done on Stenter which are drying, heat setting, finishing and coating. The Stenter machine usually consists of 8-10 chamber and each chamber contains 2-blowers, each blower provided in seperate casing and each casing contains 12 nozzles, i.e total 24 nozzles are provided in each chamber (12 facing down and 12 facing up). In each nozzle there is 48 openings, each of which are supplying the air to the fabric.

In present study, CFD analysis of fan house was carried out in three stages. In first stage, CFD analysis of Fan B with casing was done; in secound stage CFD analysis Fan A with casing was done; finally in third stage CFD analysis of fan B with nozzles was performed. Results were presented in terms pressure and velocity contours. The CFD stimulation results were compared with experiments and CFD results provided by M/s InspirOn engg. Pvt Ltd, Odhav, Ahmedabad.

- **Stage 1: CFD analysis of Fan B with casing**
  - At casing outlet non-uniform distribution of air was found which may be due to unsymmetric position of Fan in the casing,
  - Head generated , power input and efficiency of Fan B with casing were found to be 70.5 mm water column, 4.2 kW and 50% respectively.
- **Stage 2: CFD analysis of Fan A with casing**

- The methodolgy adopted in this case was similar to stage-1.
- In this case also non-uniform distribution of air was observed at casing outlet.

- **Stage 3: CFD analysis of Fan A with nozzles**
  - From analysis, the velocity at inlet and outlet of nozzle were found to be 10 m/sec and 38-40 m/sec respectively.
  - The corresponding experimental values provided by IEPL were 7.1 m/sec and 24-28 m/sec respectively, which shows quiet good agreement of the results.

6.2 Future Work

- To study the effects of various modifications on fan house performance such as: by considering symmetric position of the blower, by changing the type of blower (radial/forward), by considering the volute profile of casing etc.
- To study the effects of various modifications on nozzle performance viz. shape, pitch, size and location of nozzle openings.
- CFD analysis of complete assembly (i.e. air blower with duct passage and nozzle) in view of optimization of the Stenter machine.

Bibliography

[1] S.M. Yahya, Fundamentals of compressible flow with Aircraft and Rocket Propulsion, New age international (P) Limited, publishers, New Delhi,2007

[2] Robert D. Zucker, Oscar Biblarz, Fundamentals of gas dynamics, second edition, John Wiley & Sons, inc, 2007

[3] Carl Wassgren, Notes on Fluid Mechanics and Gas Dynamics, School of Mechanical Engineering, Purdue University, 16 Aug 2010.

[4] Breitsamter, Wake vortex characteristics of transport aircraft, ELSEVIER,30 November,2010

[5] Dr. Christian Breitsamter, Wake vortex alleviation by means of passive vortex devices, WakeNet2-Europe, Toulouse, 9-10 Feb 2005.

[6] R. Barrio, J. Fernandez, J. Parrondo and E. Blanco, "Performance prediction of a centrifugal pump working in direct and reverse mode using Computational Fluid Dynamics ", International Conference on Renewable Energies and Power Quality, 2010.

[7] Jaymin Desai, Vishal Chauhan and Shahil Charnia, Kiran Patel, "Validation of Hydraulic design of a Metallic Volute Centrifugal pump using CFD", The 11th Asian International Conference on Fluid Machinery and The 3rd Fluid Power Technology Exhibition, November 21-23, 2011.

[8] Sławomir Dykas and Andrzej Wilk, "Determination of the flow characteristic of a high-rotational centrifugal pump by means of CFD methods", task quarterly 12 no 3,2008.

[9] Lamloumi Hedi, Kanfoudi Hatem and Zgolli Ridha, "Numerical Flow Simulation in a Centrifugal Pump", International Renewable Energy Congress, November 5-7, 2010.

[10] Liu Houlin, Wang Yong, Yuan Shouqi, Tan Minggao, and Wang Kai, "Effects of Blade Number on Characteristics of Centrifugal Pumps", Chinese journal of mechanical engineering Vol. 23, 2010

[11] NPTEL courses published by IITM,Minitry of HR,government of india

[12] Hamed H. Saber,"Investigation of thermal performance of reflective insulations for different applications".Building and Environment, vol. 52, pp. 32-44, 2011

[13] Y. Gao et al,"Reduced linear state model of hollow blocks walls, validation using hot box measurements".Energy and Buildings, vol. 36, pp. 1107-1115, 2004

[14] Karim Ghazi Wakili et al,"Experimental and numerical thermal analysis of a balcony board with integrated glass fibre reinforced polymer GFRP elements".Energy and Buildings, vol. 39, pp. 76-81, 2006

[15] J.M. Sala et al,"Static and dynamic thermal characterisation of a hollow brick wall: Tests and numerical analysis".Energy and Buildings, vol. 40, pp. 1513-1520, 2008

[16] Changhai Peng, Zhishen Wu,"In situ measuring and evaluating the thermal resistance of building construction".Energy and Buildings, vol. 40, pp. 2076-2082, 2008