**Experimental Modeling of the Flow in the Furnace Top of an Ethane Cracker at EPEMSB Paka**

By

MOHAMMAD RAZISWADY BIN SALIM

FINAL REPORT

Submitted to the Mechanical Engineering Programme
in Partial Fulfillment of the Requirements
for the Degree
Bachelor of Engineering (Hons)
(Mechanical Engineering)

Universiti Teknologi Petronas
Bandar Seri Iskandar
31750 Tronoh
Perak Darul Ridzuan

# CERTIFICATION OF APPROVAL

**Experimental Modeling of the Flow in the Furnace Top of an Ethane Cracker at EPEMSB Paka**

by

Mohammad Raziswady bin Salim

A project dissertation submitted to the
Mechanical Engineering Programme
Universiti Teknologi PETRONAS
in partial fulfilment of the requirement for the
Bachelor of Engineering (Hons)
(Mechanical Engineering)

Approved:

Mr. Rahmat Iskandar Khairul Shazi
Project Supervisor

UNIVERSITI TEKNOLOGI PETRONAS
TRONOH, PERAK

November 2008

## CERTIFICATION OF ORIGINALITY

This is to certify that I am responsible for the work submitted in this project, that the original work is my own except as specified in the references and acknowledgements, and that the original work contained herein have not been undertaken or done by unspecified sources or persons.

Mohammad Raziswady bin Salim

# ABSTRACT

This report consists of the final year project entitled Experimental Modeling of the Flow in the Furnace Top of an Ethane Cracker at EPEMSB Paka. It was initialized to study a breakdown which occurred at an ethane cracking plant. The failure caused damage to the inducer fan used to induce flue gasses from the furnace to the environment. The problem was fixed but the root causes are to be determined so that history would not repeat itself and preventive maintenance could be done. In order to do so, experimental modeling using a smaller prototype is used for research work as it is cost effective and reliable. Studies were made especially in the fluid dynamics and thermodynamics properties including applications such as laminar and turbulence flows, frictional drag, the continuity equation, the ideal gas equations, and flow through non-circular conduits.

As for the modeling, certain factors have to be fulfilled to increase its reliability and to make sure that the results achieved are acceptable. Studies on dimensional analysis and the dimensional numbers that are feasible for the experiment are done particularly on the Reynolds Number and Mach number. This will enhance the similarity of the flow inside the model to the actual flow in the furnace, validating all the results achieved while experimentation took place. Since this is a continuation of previous project, some modification of the design has to be made and experimentation is conducted after the fabrication of the newly approved design has finished

Results indicate that the flow visualization varies with the damper angles and at several conditions discussed in chapter 4. However there are several limitations to achieve precise results due to time constraint and size of prototype which is 1/10 times smaller than the actual furnace top as well as experimental results that diverge from actual conditions due to assumptions made during calculations. Apart from that, through the flow visualization, induced turbulence can be detected when the flapper is rotating loosely.

## ACKNOWLEDGEMENTS

Completion of this final year project would not have been possible without the assistance and guidance of certain individuals. Their contribution both technically and mentally is highly appreciated. I would like to thank the FYP committees of The ME Department for their effort and commitment in ensuring smooth planning and scheduling of FYP lecturers, seminar ad presentations.

In particular, I would like to express my sincere and outmost appreciation to my supervisor, Mr. Rahmat Iskandar Khairul Shazi for his guidance, advice and commitment throughout the process of conducting the final year project. His concern toward the project has helped me tremendously in achieving the objectives.

I would also like to express my gratitude towards several technicians of the Mechanical Engineering department especially Mr. Jani Alang Ahmad, Mr. Zailan Alang Ahmad and Mr. Hafiz for their technical support during fabrication and experimentation.

Thanks are extended to my course mates and university mates, Nurdiana Binti Mohd Nuri, Mohd Norzahidi bin Md Yusoff, Irzan Asraf bin Abd Aziz, Ikhwan Solihin Bin Mat Zuki, Makalo Khojane and several others for their opinions and assistances in this project during fabrication and experimentation.

Last but not least, I would like to thank both my parents En Salim Bin Abdullah and Pn Faridah binti Mat for their never ending moral support from the beginning to the end of this project.

Thank You.

# TABLE OF CONTENTS

LIST OF TABLES ..... ix
LIST OF FIGURES..... x
CHAPTER 1 INTRODUCTION ..... 1
1.1 Background of Study..... 1
1.2 Problem Statement ..... 6
1.3 Objectives..... 6
1.4 Scope of Work..... 6
CHAPTER 2 LITERATURE REVIEW AND THEORY ..... 7
2.1 LAMINAR, TRANSITIONAL AND TURBULENT FLOW ..... 7
2.2 FRICTION DRAG ..... 7
2.3 DIMENSIONAL ANALYSIS ..... 8
2.3.1 Dimensional analysis and similarity..... 9
2.3.2 Reynolds Number (Re) ..... 10
2.3.3 Mach number..... 11
2.3.4 Pressure coefficient..... 12
2.4 THE CONTINUITY EQUATION..... 12
2.4.1 Venturi ..... 13
2.5 THE IDEAL GAS ..... 13
2.5.1 Properties of Ideal Gases ..... 13
2.6 FLOW IN NON-CIRCULAR CONDUITS..... 14
2.7 CRACKING FURNACE ..... 15
CHAPTER 3 METHODOLOGY ..... 16
3.1 DIMENSIONAL SIMILARITY ..... 17
Reynolds number calculation..... 17
Calculations for Velocity of Model ..... 18
Compressibility of Fluid in the Furnace Top..... 20
3.2 DESIGNING THE TEST RIG OF THE FURNACE TOP..... 21
3.3 VARIABLES..... 22
3.3.1 Flow rate into the test section ..... 22
3.3.2 Angle of damper flappers ..... 22
3.3.3 Outlet geometry ..... 23
3.4 PUMP REQUIREMENTS ..... 23

3.5 MATERIAL PROCUMENT........................................................25
3.5.1 Inventory List........................................................25
3.6 FABRICATION........................................................27
3.6.1 The test rig........................................................27
3.6.2 The Damper Flappers........................................................27
3.6.3 Flow Visualization........................................................27
3.6.4 Pressure tapings installation........................................................28
3.7 Fluid circulation system and Experimentation........................................................30
CHAPTER 4........................................................31
4.1 Results........................................................31
4.1.1 Flow visualization........................................................31
4.1.2 Pressure reading........................................................32
4.2 Discussion........................................................33
4.2.1 Flow visualization........................................................33
4.2.2 Pressure reading........................................................36
4.2.3 Limitations........................................................37
Dimensionless Parameters........................................................37
CHAPTER 5 RECOMMENDATIONS AND CONCLUSIONS........................................................40
5.1 RECOMMENDATIONS........................................................40
5.1.1 Recommendations for the furnace top........................................................40
5.1.2 Recommendations for the test rig/prototype........................................................41
5.2 Conclusion........................................................41
REFERENCES........................................................43
APPENDICES........................................................44
Appendix A TABLES........................................................46
Appendix B FIGURES........................................................49

# LIST OF TABLES

Table 3.1 : Item purchased/Obtained for Fabrication ........................................... 25

Table 3.2 : Consumables Purchased/Obtained for Fabrication ............................. 26

Table A.1: Primary dimensions and their symbols, SI, English and BG units ..... 46

Table A.2: Common quantity with their primary dimensions .............................. 46

Table A.3: Some common established nondimensional parameter or Πs encountered in fluid mechanics and heat transfer .......................................... 47

Table A.4: Physical properties of flue gases ......................................................... 48

# LIST OF FIGURES


Figure 1.1: Induction Fan ..... 2
Figure 1.2: Fan Cowling ..... 2
Figure 1.3: Cotter Pins and Linkage ..... 3
Figure 1.4: Assembly of Cotter Pin and Linkage ..... 3
Figure 1.5: Failed Cotter Pin ..... 4
Figure 1.6: Flow pattern inside the furnace caused by freely flapping damper ..... 5
Figure 3.1 : Cubic Outlet ..... 23
Figure 3.2: Position of the damper flappers and the dye injectors ..... 28
Figure 3.3 : Location of pressure tapings (blue in color) ..... 29
Figure 3.4: Manometer panel ..... 29
Figure 3.5: Experimentation setup ..... 30
Figure 4.1: Flow at angle of dampers 60 degree ..... 32
Figure 4.2: Fluid flow visualization to the top of prototype ..... 32
Figure 4.3: Condition 1 ..... 33
Figure 4.4: Sketch of the flow pattern of condition 1 ..... 34
Figure 4.5: Condition 2 ..... 34
Figure 4.6: Sketch of the flow pattern of condition 2 ..... 35
Figure 4.7: Condition 3 ..... 35
Figure 4.8: Sketch of the flow pattern of condition 3 ..... 36
Figure 4.9: Vectors with a fan ..... 38
Figure 4.10: Vectors without a fan ..... 38
Figure 5.1 Design modification of the furnace top ..... 40
Figure B.1: Side view drawing of the actual furnace at EPEMSB Paka ..... 49
Figure B.2: Front view drawing of the actual furnace at EPEMSB Paka. ..... 50
Figure B.3: Front view of the prototype (Dimensions are in millimeter, mm) ..... 51
Figure B.4: Side view of the prototype ..... 52
Figure B.5: Plan view of the prototype ..... 53
Figure B.6: Oblique drawing (3D) of the prototype ..... 54
Figure B.7: Moody Chart ..... 55

# CHAPTER 1
# INTRODUCTION

## 1.1 Background of Study

Ethylene Polyethylene Malaysia Sdn Bhd (EPEMSB) is a petrochemical plant producing polyethylene as their main output. Using a method called ethane cracking [10]; the plant uses a furnace to discharge what is known as flue gas from the process to outdoor environment. This furnace is equipped with an induction fan (Figure 1.1) to create artificial drought enhancing the discharge of the flue gas. On 3 September 2005, the induction fan of the ethane cracking furnace F-102 at EPEMSB Paka have failed along with its cowling (Figure 1.2) suspected due to high vibrations as a result from a broken cotter pin (Figure 1.3 and Figure 1.5) on one of the four damper flappers. Due to the failed flapper, the flow pattern of the artificial draught was distorted (Figure 1.6) causing an induced turbulence and hence vibrations well above danger level. This event has affected daily production which led to major losses in production and capital investment. However, there was no direct visualization on what actually happened inside the furnace. In order to proof the root cause of the failure, experimental modeling is used since the actual furnace is not available for studying purpose. This is a very good way in solving mostly problems occurring in plants. It is safer, cheaper and more environmental friendly. However, the challenge is to design a model that could satisfy most of the properties that will be discussed further in the literature review within this report.

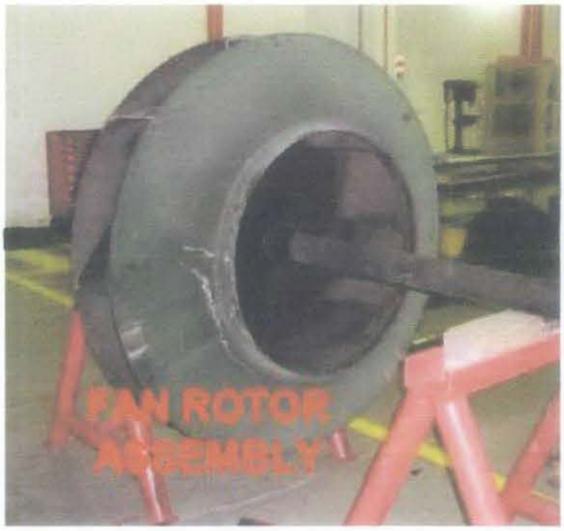


**Figure 1.1: Induction Fan**


**Figure 1.2: Fan Cowling**

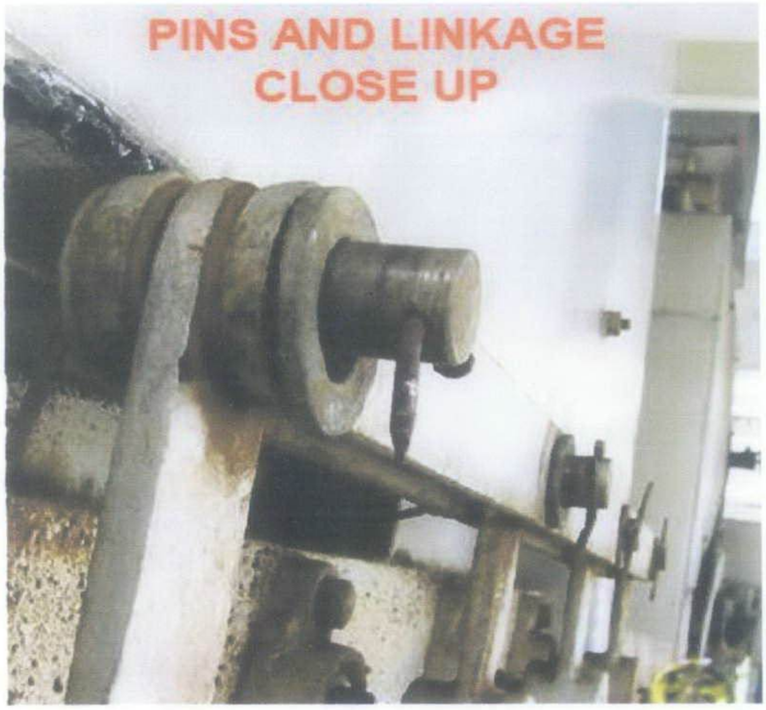


**Figure 1.3: Cotter Pins and Linkage**

**Figure 1.4: Assembly of Cotter Pin and Linkage**

**Figure 1.5: Failed Cotter Pin**

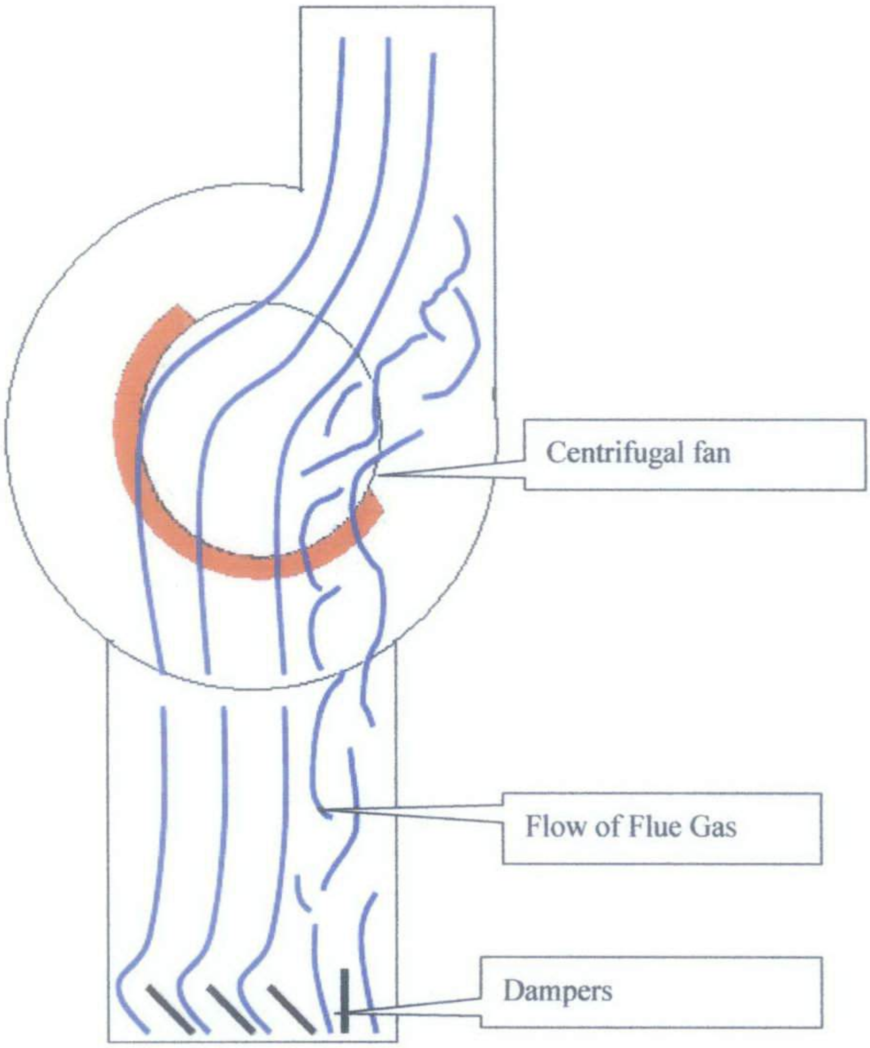


**Figure 1.6: Flow pattern inside the furnace caused by freely flapping damper**

## 1.2 Problem Statement

It is impossible to stop daily production just to use the furnace for troubleshooting. Leaving with no other alternatives, production has to be stop for quite a while since the fan needs to be replaced and it cannot be done in a days work. By doing so, EPEMSB looses a lot of money because their operation was halted. There is also the cost to replace the fan. Such liability can greatly affect the plant's capital and they surely do not want the tragedy to repeat itself. Such accident can also result in injury or even fatality which is worst that ever. In order to prevent such accident, the root cause of the failure without having to stop production has to be determined. Therefore experimental modeling is suggested for its advantages.

## 1.3 Objectives

- Modifying the existing furnace top model
- Conduct an experiment to visualize the flow in a furnace top. Analyze and visualize the flow inside the furnace, especially the part where the flappers are positioned.
- Postulate the possible causes of turbulence.

## 1.4 Scope of Work

Literally, the project will cover laminar and turbulent flows, friction drag, and losses in pipes, dimensional analyzing, and the continuity equation of fluid flow, the ideal gasses and the flow visualization using screening. This experiment also requires calculation velocity model (dimensional analysis), flow rate of fluid and calculations for drivers and pumps requirement and material selection. Furthermore, a prototype/test rig will be design and fabricated based on the analysis done. Experimental work flow visualization will also be commencing. Eventually results from the experiments will be analyzed to postulate the major cause of turbulence inside the furnace.

# CHAPTER 2
# LITERATURE REVIEW AND THEORY

This section will discussed all the studies and research done which is related to this project. Most of the properties discussed are to be understood for they are used as basis for experimental modeling.

## 2.1 LAMINAR, TRANSITIONAL AND TURBULENT FLOW

There are basically two types of flows which are **laminar** and **turbulence**;

Fluid flow that is slow tends to be **laminar**, as it speeds up a **transition** occurs and it crinkles up into complicated, random **turbulent** flow. To vary different types of flow, they are separated using Reynolds Numbers. A laminar flow is when a fluid flow is in parallel layers, with no disruption between layers. Laminar flow is a flow regime characterized by high momentum diffusion, low momentum convection, pressure and velocity independent from time. Transitional is the state where a flow is changing from a laminar to turbulence flow. Here the flow displays a general pattern, however, with several random disorders along the flow line. Turbulent flow is in a disordered pattern. It is a flow regime characterized by chaotic form. This includes low momentum diffusion, high momentum convection and rapid variation of pressure and velocity in space and time. In turbulent flow, unsteady vortices appear on many scales and interact with each other. The drag, due to skin friction, is relatively high.

From the problem statement, the fan was assumed to be damaged due to vibration created by turbulence in the flow inside the furnace. There will be a follow up for studies in turbulence once the experiment can be modeled.

## 2.2 FRICTION DRAG

All moving object will experience drag. In moving fluid, drag occurs between the surface of the moving fluid and to whatever surface it made contact with. This is a combination of flow direction components of the normal and tangential forces on the body. Typically, the result for a given shaped object is a drag coefficient, $C_D$, where:

$$C_D = \frac{D}{\frac{1}{2}\rho U^2 A} \qquad \text{(Eq. 2.1)}$$

Friction drag, $D_f$, is that part of the drag that is due directly to shear stress, $\tau_w$, on the object. Friction drag depends on the orientation of the body as well as the magnitude of the wall shear stress $\tau_w$. Commonly, friction drag act as a reducer to an actual Reynolds number of a flow. During the experimentation, friction drag acted on the damper flappers. It increases as the angle of the flappers are increased causing higher amount of pressure at the flappers.

## 2.3 DIMENSIONAL ANALYSIS

As being stated before, to especially save time and money, geometrically scaled models are used for modeling and tests than full-scale prototype. According to Cengel and Cimbala [1], we have to be very careful in scaling a model. This is where dimensional analysis plays its role. It is very useful especially when it is necessary to conduct experiment or analysis.

It is as understood that in general flow field, complete similarity between model and prototype is achieved only when there is geometric, kinematic, and dynamic similarity.

There are seven primary dimensions which is also known as fundamental basic of dimensions which are – mass, length, time, temperature, electric current, amount of light, and amount of matter. (Refer Appendix A – Table A.1 and

Table A.2)

### *2.3.1 Dimensional analysis and similarity*

This powerful **technique** has three primary purposes:

- Generate non-dimensional parameters that help in the design of experiments (physical/numerical) and in the reporting of experimental results. [1]
- To obtain scaling laws so that prototype performance can be predicted from model performance [1]
- To (sometimes) predict trends in the relationship between parameters. [1]

Before the author could consider dimensional analysis, he should be aware of three necessary conditions for complete similarity between model and prototype.

**Geometric Similarity** - the model must be the shape as the prototype, but may be scaled by some constant scale factor.

**Kinematic Similarity** – the velocity at any point in the model flow must be proportional (constant scale factor) to the velocity at the corresponding point in the prototype flow. [1]

**Dynamic Similarity** - achieved when all forces in the model flow scale by a constant factor to corresponding forces in the prototype flow (*Force - scale* equivalence)

A dimensional parameter is denoted as Pi (**Π**). A dependent Π is noted as $\Pi_1$ which then becomes a general function to several other Πs'. In order to achieved complete similarity, besides making sure of geometrical similarity, all independent Π groups must match between model and prototype. [1]

Several methods were developed to enhanced dimensional analysis and the most popular method is the method of repeating variables popularized by Edgar Buckingham (1867-1940). It is also known as Buckingham Pi Theorem. (Refer Appendix A –

Table **A.3**)

Throughout the author's observation, also based on previous researches regarding this project, several dimensional group or Πs are to be generated. These parameters will then aid in achieving dimensional similarity. [1]

### 2.3.2 *Reynolds Number (Re)*

Another important group that will aid in this project is the dimensionless group that expresses the dynamic properties of fluid. Such group is known as the Reynolds number, Re and is define as:

$$R_e = \frac{VD\rho}{\mu} = \frac{VD}{v} \qquad \text{(Eq. 2.2)}$$

*where,*

*Re = Reynolds number*

*V = velocity of the fluid*

*D = diameter/hydraulic diameter*

*$\rho$ = density*

*$\mu$ = kinematics viscosity*

The Reynolds number acts as an indicator whether the flow over surface is laminar or turbulence. When Reynolds number is small flow is laminar; when it is large, turbulent. It is stated that whenever a flow changes from laminar to turbulent, the functional relation between the Nusselt and Reynolds numbers changes considerably.

There is a principle of fluid dynamics known as the 'model law' which states the following (according to Kreith, 1966): "The behavior of two systems will be similar if the ratios of their linear dimensions, forces, velocities, etc, are the same. Under condition of forced convection for geometrically similarities systems, the velocity field will be similar provided the ratio of inertial forces to viscous forces is equal in both fluids. Being the ratio of these forces, we can expect similar flow conditions for a given value of Reynolds number. [1] By assuming that the Reynolds number is the same for the actual and prototype of the furnace, the author manage to calculate the velocity of the flow at the inlet and outlet of the prototype thus enables him to calculate the flow rate needed to match the actual condition.

### *2.3.3 Mach number*

**The Mach Number** is a dimensionless value useful for analyzing fluid flow dynamics problems where compressibility is a significant factor. It is the ratio of the velocity (V) of the fluid (alternatively the velocity of a body immersed in a fluid) to the speed of sound (c) in the fluid. It will determine the compressibility of fluid used in the experiment. (Incompressible fluid has no time dependant varying density gradient) [1]

$$Ma = \frac{V}{c} \qquad \text{(Eq. 2.3)}$$

*Where,*

*Ma = Mach number*

*V = velocity*

*C = speed of sound*

From the calculation in the methodology, the flow of flue gas inside the actual furnace is at Ma< 0.3. This means that the flow is incompressible thus the fluid motion has negligible changes in density. In incompressible flow, an increase in velocity is associated with a decrease in the cross-sectional area of the duct which then leads us to the venturi effect. [1]

### 2.3.4 *Pressure coefficient*

The pressure coefficient is a non-dimensional parameter which consists of the static pressure over the dynamic pressure of the fluid. It is govern by the equation

$$C_p = \frac{p - p_x}{\frac{1}{2}\rho U^2} \qquad \text{(Eq. 2.4)}$$

*where,*

$C_p$ = *pressure coefficient*

$p$ = *pressure*

$p_x$= *atmospheric pressure*

$\rho$ = *density*

$U$ = *velocity*

However, the author could not acquire the actual pressure at the inlet and outlet pressure which is required to determine the pressure values at the prototype.

## 2.4 THE CONTINUITY EQUATION

Every type of fluid is considered to have mass, and where there is a mass going in, there have to be a mass going out. This is govern by the conservation of mass which states that

$$\sum \dot{m}_{in} = \sum \dot{m}_{out} \qquad \text{(Eq. 2.5)}$$

$$(\rho A V)_{in} = (\rho A V)_{out} \qquad \text{(Eq. 2.6)}$$

Here, we can see that for a control volume, the mass flow rate of the fluid into the control volume must be equal to the mass leaving the control volume. [2]

By assuming that the flow density remained constant, we can simplify above equation into

$$Q = AV_1 = AV_2 \qquad \text{(Eq. 2.7)}$$

*where,*

$Q$ = *flow rate of fluid*

$A = cross\ sectional\ area$

$V = velocity\ of\ flow$

This is what we called the **continuity equation** for steady one-dimensional incompressible flow.

### *2.4.1 Venturi*

To be exact, a venturi is the effect of fluid flow through different types of control volume which much satisfy the continuity equation (Eq. 2.7) for incompressible flow. Fluid velocity varies inversely with the cross sectional area. For instance, if the fluid is moving from a bigger area through a throat (smaller area), the fluid will accelerate and vice versa. The cross sectional area of the prototype varies at both the inlet and exit. There is also a reservoir fixed under the prototype which also has a different area that should be considered to maintain the flow rate of fluid into the prototype. [11]

## 2.5 THE IDEAL GAS

### *2.5.1 Properties of Ideal Gases*

Most normal gases at normal pressures and temperature can be treated as ideal gases provided that there are not phase changes occurring.

The equation of state for an ideal gas provides a relationship among temperature (T), pressure (P), and the specific volume (v). The most convenient form of the ideal gas equation of state is:

$$\mathbf{RT = Pv} \qquad \text{(Eq. 2.8)}$$

Where R is the gas constant for the particular of gas of interest and is given by

$$R = \frac{R_u}{MW} \qquad \text{(Eq. 2.9)}$$

Where MW is the molecular weight of the particular gas and $R_u$ is the universal gas constant. Another way to express the ideal gas equation is:

$$\text{Ideal Gas Law: } PV = nRT = NkT \qquad \text{(Eq. 2.10)}$$

where:

n = number of moles

$$R = \text{universal gas constant} = 8.3145 \ \frac{\mathrm{J}}{\mathrm{mol \cdot K}}$$

N = number of molecules

$$k = \text{Boltzmann constant} = 1.38066 \text{ x } 10\text{-}23 \ \frac{\mathrm{J}}{\mathrm{K}} = 8.617385 \text{ x } 10\text{-}5 \ \frac{\mathrm{eV}}{\mathrm{K}}$$

$$k = \frac{R}{NA}$$

$$NA = \text{Avogadro's number} = \frac{6.0221X10^{23}}{mol}$$

The ideal gases properties also states that there are no attractive or repulsive forces between the molecules or between the molecules and the walls of the container and all collisions of molecules are perfectly elastic. Therefore, all the internal energy is in the form of kinetic energy and any change in internal energy is accompanied by a change in temperature. Since the experiment is done at constant temperature, so there are no change in the kinetic energy and internal energy inside the prototype. [8, 9]

## 2.6 FLOW IN NON-CIRCULAR CONDUITS

For the relationship between the relative surface roughness, friction factor and the Reynolds number of a flow on a surface, engineers often use moody chart (Figure B.8).However this is only practical with flow through circular conduits. Regarding the test rig, it consists of several components which most of them are not circular in geometry. Therefore, several adjustments to the calculation of Reynolds number and such have to be made to achieve similarity to the actual condition. Several authors have agreed on the hydraulic diameter, defined as four times the ratio of the cross sectional flow area divided by the wetted parameter of the non circular conduit. The hydraulic diameter, $D_h$ is used to geometrically modify the parameters of the conduit to ensure that the flow kinematics (average velocity) would be the same as that of a flow in a circular conduit with the equivalent diameter, $D_{eq}$

## 2.7 CRACKING FURNACE

Ethane, $C_2H_6$ is obtained from natural gasses field and as by product of petroleum refining. Being the second largest natural gas source, ethane is chemically used in the production of polyethylene. Ethane is being diluted with steam and heated to very high temperature (more than $900^0C$). [10]

At this temperature, heavy hydrocarbon molecules are broken down to smaller and lighter hydrocarbon molecules, where saturated hydrocarbons become unsaturated. This process is called Cracking process [10]. As mentioned before, this process is used to produces lighter hydrocarbon molecules from heavy hydrocarbon molecules whereas in this case, we obtained ethylene (light hydrocarbon molecules) from ethane (heavy hydrocarbon molecules).

The rate of cracking is dependant to the temperature and the presence of catalyst. This process can also be referred to as pyrolysis, the breaking of bigger alkane to smaller and useful alkanes or alkenes. The heating took place without any presence of oxygen. Normally, the reaction temperature is very high but it is only allowed for a short period of time. The gasses are then quickly quenched out of the furnace to stop the reaction. Note that these gasses are being basically flue gasses. (Appendix A - Table A.4)

# CHAPTER 3
# METHODOLOGY

In order to achieve the goal of this project, the problem statement and objectives will be defined first. The main objective of this project is to experimentally study the flow that is occurring inside the furnace and finding the solution to solve the problem statement. There is also an initial objective that has to be fulfilled which is modifying the existing prototype to match the condition and properties of the actual furnace at EPEMSB Paka.

At first all the necessary literature reviews were studied. Reading consists of previous experimentation [8, 9], journals [10] books [1, 2] and several engineering websites. Next the author focused on dimensional analysis which consist of the velocity calculation, flow rate calculation and translating the actual drawings to a scaled down prototype. From there, AutoCAD software is used to generate a new design which is considered to consist of the main part for the flow in the actual furnace. It will show all the necessary engineering drawing of the test rig. (Figure B.3, Figure B.4, Figure B.5, Figure B.6 and Figure B.7 ) The next step was fabricating the test rig based on the analysis done previously. Gaskets and filling were used between the compartments to prevent leakage. After the fabrication was finished, experiments and modeling were done in the lab to achieve the main objective of this project.

Eventually an analysis was done to interpret the results achieve during experimentation and the possible cause of turbulence was postulated.

## 3.1 DIMENSIONAL SIMILARITY

The model has been scaled down to 1/10 of the original furnace top geometry. As with many scaled down test rigs involving fluids, a suitable medium must be used as a substitute for air in the original furnace top. From the calculations generated below, water, with its very low kinematic viscosity, will generate an inlet and outlet velocity of **3.976 m/s and 7.044 m/s** respectively. To obtain dimensional similarity, the Reynolds number value has to be the same for the model and prototype. From the similarity the velocity profile of the prototype is calculated. The model cannot be scaled down any further to avoid higher velocities that would make it impossible for flow visualization and may cause the test section to fail due to high pressure.

***Reynolds number calculation***

Volumetric flow rate, $Q_{(model)}$ $= 15.4\ m^3/s$

Kinematic Viscosity$_{(model)}$, $\upsilon_{flue} = 3.28 \times 10^{-5}\ m^2/s$ (Flue gas at 200° C) [Table A.4]

***Inlet;***

Inlet Area$_{(model)}$, $A_{inlet}$ $= 1.15\ m \times 1.15\ m$

$= 1.3225\ m^2$

Inlet perimeter$_{(model)}$, $P_{inlet}$ $= 1.15 + 1.15 + 1.15 + 1.15$

$= 4.60\ m$

Inlet Equivalent Hydraulic Diameter

Diameter$_{(model)}$ , $D_{inlet}$ $= 4A / P$

$= 4(1.3225)\ m^2 / 7.2m$

$= 1.15\ m$

$Q_{inlet(model)}$ $= V_{inlet} \times A_{inlet}$

$V_{inlet(model)}$ $= Q_{inlet}/A_{inlet}$

$= 15.4\ m^3/s / 1.3225m^2$

$= 11.645\ m/s$

**Reynolds Number, $Re_{(model)}$** $= V_{inlet} \times D_{inlet} / \upsilon_{flue}$

$= 11.645\ m/s \times 1.15\ m / 3.28 \times 10^{-5}\ m^2/s$

$= \mathbf{408{,}285.061}$

***Outlet;***

Outlet Diameter$_{(model)}$, $D_{outlet}$ $= 0.975\ m$

Outlet Area$_{((model)}$, $A_{outlet}$ $= \pi\ (0.975\ m)^2 / 4$

$= 0.7466\ m^2$

$Q_{outlet(model)}$, $= V_{outlet} \times A_{outlet}$

$V_{outlet(model)}$, $= Q_{outlet} / A_{outlet}$

$= 15.4 m^3/s / 0.7466\ m^2$

$= 20.63\ m/s$

**Reynolds Number, $Re_{(model)}$,** $= V_{outlet} \times D_{outlet} / \upsilon_{flue}$

$= 20.63\ m/s \times 0.975\ m / 3.28 \times 10^{-5}\ m^2/s$

$= \mathbf{613{,}239.33}$

***Calculations for Velocity of Model***

**With a model to prototype geometric ratio = 1:10 and water as the working fluid*

Kinematic Viscosity$_{(prototype)}$, $\upsilon_{water}$ $= 1.12 \times 10^{-6}\ m^2/s$ (Water at 15.6° C)

***Inlet;***

Inlet Area$_{(prototype)}$, $A_{inlet}$ $= 0.115\ m \times 0.115\ m$

$= 0.013225\ m^2$

Inlet perimeter$_{(prototype)}$, $P_{inlet}$ $= 0.115\ m + 0.115\ m + 0.115\ m + 0.115\ m$

$= 0.460$ m

Equivalent Hydraulic

**Diameter$_{(prototype)}$, D$_{inlet}$** $= 4A/P$

$= 4 \text{ X } 0.0324 \text{ m}^2 / 0.72\text{m}$

$= 0.115$ m

**Reynolds Number, Re$_{(model)}$\= Re$_{(prototype)}$** = 408,256.061

*To achieve the same Reynolds Number;*

Re$_{(prototype)}$ (inlet) = Re$_{(model)}$(inlet) $= V_{inlet} \text{ X } D_{inlet} / \upsilon_{water}$

$= V_{inlet} \text{ X } 0.115 \text{ m} / 1.12 \text{ X } 10^{-6} \text{ m}^2/\text{s}$

Inlet Velocity$_{(prototype)}$, V$_{inlet}$ $= \text{Re X } \upsilon_{water} / 0.115$ m

= **408,256.061** $\text{X } 1.12 \text{ X } 10^{-6} \text{ m}^2 / 0.115$ m

$= 3.976$ m/s

***<u>Outlet;</u>***

Outlet Diameter$_{(prototype)}$, D$_{outlet}$ $= 0.0975$ m

Outlet Area$_{(prototype)}$, A$_{outlet}$ $= \pi (0.0975 \text{ m})^2 / 4$

$= 0.007466 \text{ m}^2$

**Reynolds Number, Re$_{(model)}$\= Re$_{(prototype)}$ = 613,239.33**

*To achieve the same Reynolds Number;*

**Re$_{(model)}$(outlet)\=Re$_{(prototype)}$(outlet)** $= V_{outlet} \text{ X } D_{outlet} / \upsilon_{water}$

$= V_{outlet} \text{ X } 0.0975 \text{ m} / 1.12 \text{ X } 10^{-6} \text{ m}^2/\text{s}$

Model Outlet Velocity$_{(prototype)}$, V$_{inlet}$ $= \text{Re X } \upsilon_{water} / 0.075$ m

= **613,239.33** $\text{X } 1.12 \text{ X } 10^{-6} \text{ m}^2 / 0.0975$ m

$= 7.044$ m/s

Since the inlet and outlet velocities of the furnace top were unavailable, the author used the continuity equation based on the assumptions made above, and computed the outlet velocity from the flow rate and area measurements provided. An incompressible flow was confirmed once the Mach number indicated that the flow at both inlet and outlet of the furnace top was incompressible at 0.0479 (compressible at Ma> 0.3 [1]). The calculation for the Mach number is as follows;

***Compressibility of Fluid in the Furnace Top***

Mach number, Ma $= V/a$

Speed of Sound in Air, a $= (\gamma RT)^{½}$ where $R = 286 m^2/s^2.K$ (Gas Constant)

$T$ = Absolute Temperature

Ratio of Specific Heats, $\gamma$ $= C_p/C_v$ where $C_p$ = Specific Heat (Const. Pressure)

$C_v$ = Specific Heat (const. Volume)

$C_p$ at 200°C given in table 4.1 as 1.097 kJ / kg.K

*Assumption: The molecular weight of Nitrogen gas is used as it constitutes 76% of the flue gas composition.*

1.097 kJ / kg . K X 28 kg / kmol = 30.716 j /mol . K

$C_v$ $= C_p - R$ where $R = 8.314$ J / mol . K

$= 30.716 - 8.314$

$= 22.4015$ J /mol . K

$\gamma$ $= C_p / C_v$

$= 30.716 / 22.402$

$= 1.371$

a $= (\gamma RT)^{1/2}$

Where $R = 286 m^2/s^2 . K$

And $T = (200 + 273.15)$ °K

$= (1.371 \text{ X } 286 \text{ X } 473.15)^{1/2}$

$= 430.726$ m / s

Ma $= V/a$

At Furnace Top Inlet, Ma $= 11.645 / 430.726$

$= 0.0270$

At outlet, Ma $= 20.63/ 430.726$

$= 0.0479$

*Thus the fluid is considered to be incompressible in the furnace.*

### 3.2 DESIGNING THE TEST RIG OF THE FURNACE TOP

The test rig was designed using Computer Aided Software (CAD) code AutoCAD for the isometric and orthographic drawings (Figure B.4, Figure B.5 , Figure B.6 and Figure B.7). The drawing is based on the engineering drawing of the actual furnace at EPEMSB Paka (**Error! Reference source not found.**and Figure B.3). The model was scaled down to 1/10 of the original geometry of the furnace top with a maximum height of 0.375 m and maximum combined length of 0.223 m. The dimensions of the inlet for the model are 0.115 m X 0.115 m. The damper flappers are 0.04 m in height, pivoted to the shaft at the middle, spread equivalently along the width of the base compartment. Geometry of the outlet is circular with a diameter of 0.0975 m.

### 3.3 VARIABLES

There were three variables with which the experiment was conducted:

- Flow rate into the test section
- Angle of damper flappers
- Outlet geometry

#### *3.3.1 Flow rate into the test section*

A flow control valve was positioned just before the inlet of the test rig to control the volume of working fluid into the test section. The amount of flow into the test rig as measured by the number of turns made on the gate valve controller ranging from a minimum of 0.000 $m^3/s$ to 0.0526 $m^3/s$. However the pump used/available can only deliver a maximum flow rate of 50liter per minute or 0.00083$m^3/s$. Comment on the pump are available in the experimentation section. The flow rate into the prototype is calculated based on the equation flow rate:

$$\mathbf{Q = A \times V} \qquad \text{(From Eq. 2.1)}$$

Where A = inlet area of the prototype

V = velocity of the fluid flow at the inlet of the prototype

The area of the prototype is acquired from the engineering drawing of the furnace courtesy of EPEMSB Paka. (Scaled to 1/10 of the actual furnace) and the velocity is achieved from the velocity calculation shown in section 3.1.Thus;

$$\text{Flow rate required}_{(\text{prototype})}(\text{inlet}), Q = A \text{ X } V$$

$$= 0.013225 \text{ m}^2 \text{ X } 3.976 \text{ m/s}$$

$$= \mathbf{0.0526} \text{ m}^3/\text{s}$$

#### *3.3.2 Angle of damper flappers*

At the inlet, the angle of the damper flappers opening was varied to view the flow profile. The angle of each damper flapper can be altered to view the fluid flow past the immersed metal plates with increasing angles. Each metal plate can be turned independently to a required angle which varies from 0°, 30°, 60° and 90°.

### *3.3.3 Outlet geometry*

The outlet of the furnace is fabricated in a cubic shape to match the actual furnace drawing acquired by EPEMSB Paka.

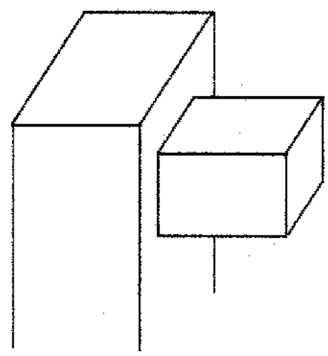

**Figure 3.1 : Cubic Outlet**

## 3.4 PUMP REQUIREMENTS

A driver provided the head to supply the water (working fluid) with enough pressure to overcome the gravitational, velocity, friction heads along with the losses through the diffuser compartment, bends and nozzles. As for the flow rate, ideally the best pump would be an axial type as it is best suited for low pressure and high volume flows. The test rig requires a discharge pressure and flow rate supply of 0.4 bars and 0.0526 $m^3/s$. The flow rate is from the calculation in section 3.3.1 while the pressure value is obtained from the total losses calculation as follow;

Total Losses

$$\frac{V_1^2}{2g} + \frac{P_1}{\gamma} + H_p = Z_2 + \frac{V_2^2}{2g} + \frac{P_2}{\gamma} + H_f$$

Equation 3.1

where,

$Z_1$ = height of the test rig base from ground level, $\approx 0$ m

$Z_2$ = total elevation from ground level (m)

$V_1$ = inlet velocity (m/s)

$V_2$ = outlet velocity (m/s)

g = gravity (m/s$^2$)

$P_1$ = inlet pressure (atmosphere)

$P_2$ = outlet pressure (atmosphere)

$H_f$ = Frictional losses

$h_M$ = major losses $\approx 0$ m (Perspex; negligible surface roughness)

$h_L$ = minor losses (m)

$H_p$ = head added by pump (m)

$Z_2 - Z_1$ = 0.375 m – 0m = **0.375 m**

$$\frac{V_2^2 - V_1^2}{2g} = \frac{(7.044\ \text{m/s})^2 - (3.976\ \text{m/s})^2}{2 \text{ X } 9.81} = \mathbf{1.723\ m}$$

$P_1 - P_2$ = **0**

$h_L$ $= K_L V^2 / 2g$ $K_L$ = 0.8 (sharp edge nozzle at outlet)

$= (0.8 \text{ X } 7.044^2) / 2g$

$= 2.02$ m

$H_f$ $= \sum h_M + \sum h_L = 0\text{ m} + 2.02\text{ m} = 2.02\text{ m}$

$H_p$ = 0.375 m 1.723 m + 0 + 2.02 m = **4.118 m** (to overcome losses)

Discharge driver pressure to overcome losses = (4.118 X 999 X 9.81) N/ m$^2$

$= 40{,}357.2 \text{ N/ m}^2$

$= 40{,}357.2 \times 1 \times 10^{-5}$ bar

$= 0.4036 \approx \mathbf{0.4\ bar}$

## 3.5 MATERIAL PROCUMENT

The main criteria for determining the materials required to build the test rig include;

- transparency for clear flow visualization
- Availability and at a reasonable price.
- have strong edges and corners to sustain under high pressure water,
- capable to measure pressure drops across the test section.(installation of pressure taps)

Considering all the factors mentioned, Perspex is used as the main material for the prototype. The material is cut and sized using the facilities such as CNC milling machine, Lathe/Turning machine and conventional milling machine that are available in the campus.

### *3.5.1 Inventory List*

The following are the lists of materials and consumables items along with the units and cost that were used to during fabrication. Some units are left without any cost as the materials/tools were available in campus with no charge while others were receipts from hardware vendors.

**Table 3.1 : Item purchased/Obtained for Fabrication**

| Items | Size | Quantity | Unit Cost | Total |
|---|---|---|---|---|
| Perspex | 2.0m X 1.0m X 0.008 m | 1 | - | - |

| | | | | |
|---|---|---|---|---|
| Fasteners | - | 18 | - | - |
| Aluminum Plate | 114mm X 40mm X 3mm | 4 | - | - |
| Aluminum Shaft | 25 mm ø X 30mm | 8 | - | - |
| Dye Injector Set | - | 1 | - | - |
| Dye Bottle | 500ml | 1 | - | - |
| Tubes | 8mm ø X 4000mm | 1 | - | - |
| O-Ring | 15mm ø | 9 | RM 0.50 | RM 4.50 |
| Centrifugal Pump | - | - | - | - |
| Metal washer | 4.5mm ø | 18 | - | - |
| PVC Pipe | 25mm ø X 2500mm | 1 | | RM 7.00 |
| PVC Connectors | - | 4 | RM 2.00 | RM 8.00 |
| Pipe Tape | | 3 | RM 0.50 | RM 1.50 |
| Flow Control Valve | 1 " | 1 | RM 10.00 | RM 10.00 |
| PVC Hexagonal Socket | 1" | 2 | RM 1.60 | RM 3.20 |
| TOTAL | | | | RM 34.20 |

**Table 3.2 : Consumables Purchased/Obtained for Fabrication**

| Consumables | Size | Quantity | Unit Cost | Total |
|---|---|---|---|---|
| Silicon Glue | 400ml | 2 | RM 6.00 | RM 6.00 |
| Chloroform | 300ml | 1 | RM 4.00 | RM 4.00 |
| PVC Glue | | 1 | - | Rm3.50 |
| TOTAL | | | | RM13.50 |

## 3.6 FABRICATION

### *3.6.1 The test rig*

The prototype was fabricated by the author using manufacturing tools and equipment such as conventional milling machine, lathe/turning machine along with Computational Numerical Control (CNC) machines available in the university. The author used chloroform to as glue to attach the Perspex together. There were also some screws and bolts used so that the prototype can be detached for modification or further enhancement.

### *3.6.2 The Damper Flappers*

The flappers were made of thick 3mm aluminum sheet cut to a size of 40mm X 114mm. These flappers were attached to aluminum shaft at each end to the side of the prototype. However, the flappers were not attached to each other but were leave to turn independently from 0°, 30°, 60° and 90°.

### *3.6.3 Flow Visualization*

The visualization kit is made from a tube connected to two dye injectors. It is placed above the dampers through a hole that is drilled on each wall side of the prototype. The dye is pumped manually into the bottle, enabling the flow pattern to be visualized from the side of the furnace prototype.

**Figure 3.2: Position of the damper flappers and the dye injectors**

### *3.6.4 Pressure tapings installation*

Five holes were drilled through the walls of the prototype to insert the pressure tapings that will be used to measure the pressure at certain points during the experimentation. Each point is chosen by assuming that there will be a significant pressure variant during experimentation. The pressure taps are connected to a manometer that is available in the lab in order to get the pressure reading.

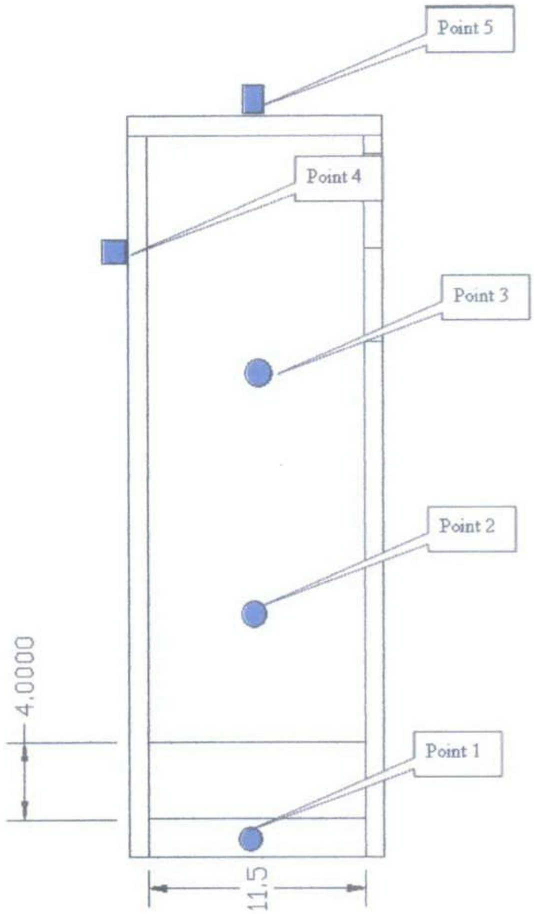


**Figure 3.3 : Location of pressure tapings (blue in color)**

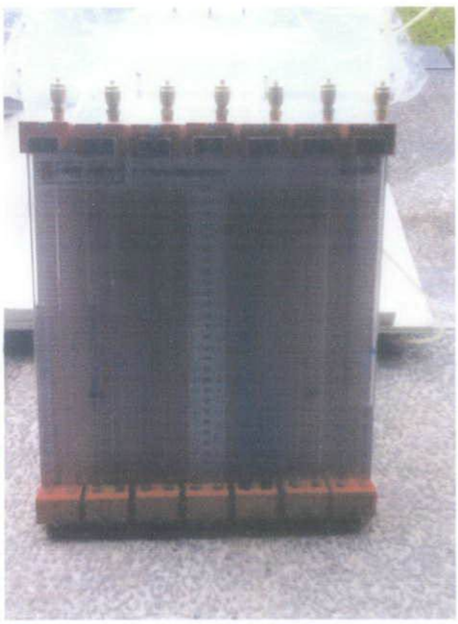

**Figure 3.4: Manometer panel**

### 3.7 Fluid circulation system and Experimentation.

With a series of fittings including the flow control valve, the water from the pump outlet was channeled into the supplementary base compartment and consequently into the test section. Once the working fluid reached the top of the test section, it will be exited out through the outlet as shown in Figure 3.5.

The experimentation commenced once fabrication was completed in Week 13. Fabrication was carried out successfully and very minimal problems were faced during testing and experimentation. The most severe drawback, however, was the pump.

**Figure 3.5: Experimentation setup**

# CHAPTER 4

## RESULTS AND DISCUSSION

### 4.1 Results

As mentioned before, the pump used can only provide a maximum flow rate of 50liter/minute. Using a valve as shown in the experimentation configuration, the flow is controlled to several flow rates which are 0.21 liter/second, 0.42 liter/second, 0.63 liter/second and a maximum of 0.83 liter/second.

#### *4.1.1 Flow visualization*

At the flow rate of 0.21 liter/second, the dye used dissipates slowly. The flow can be visualized quite clearly. However, as the flow rate is increased, the dye across the dampers dissipates at a faster rate making the flow hard to visualize.

There are several damper angle tested with three condition which are

- Angle of dampers at 0°, 30°, 60° and 90° with all four dampers are in the same position
- Angle of dampers at 0°, 30°, 60° and 90° with three dampers secured and one damper is at a different angle.
- Angle of dampers at 0°, 30°, 60° and 90° with three dampers secured and one damper is flapping.

The visualization is recorded and can be referred in the DVD-R.

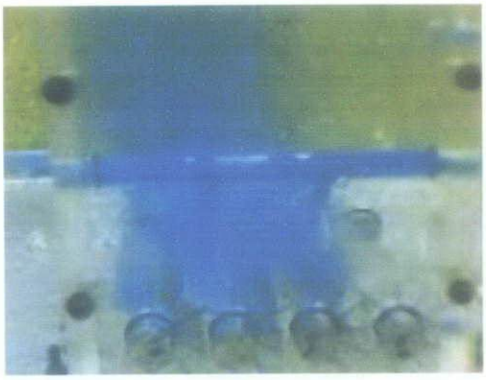

**Figure 4.1: Flow at angle of dampers 60 degree**

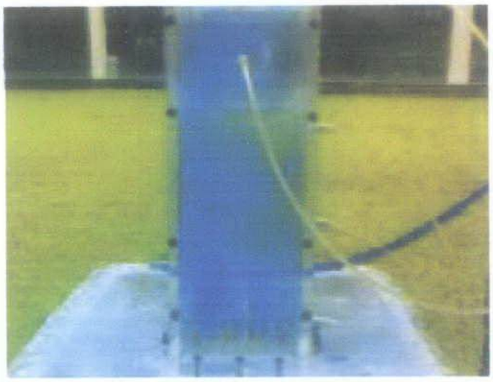

**Figure 4.2: Fluid flow visualization to the top of prototype**

### *4.1.2 Pressure reading*

There is a drawback in getting the pressure reading at all five points (Figure 3.3). This is due to insufficient scale with the manometer used. As soon as the water filled the prototype, the pressure reading at all points were already out of scale. Because of this, the author could not get any reading from all points. However the pressure patterns due to the angle of dampers are anticipated and will be elaborated in the discussion (section 4.2).

## 4.2 Discussion

### *4.2.1 Flow visualization*

The flow visualization varies with the damper angles and condition mentioned in section 4.1.1.

**Condition 1**: All four dampers secured

When all the dampers are secured at an angle of 0°, the flow moves smoothly to the top of the furnace, however when the dampers are in 30° and 60° the flow is being directed to the side wall of the furnace and the rate of the flow to reach the top increases with time. The rate also increases as the angle of the dampers increased.

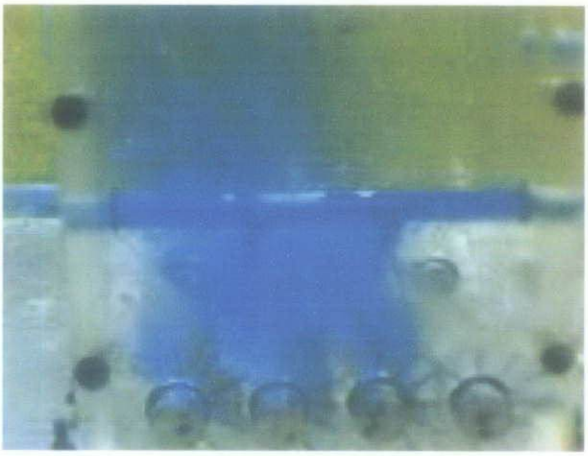

**Figure 4.3: Condition 1**

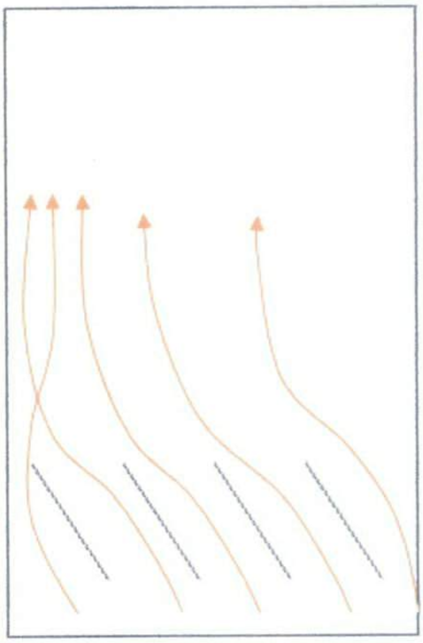

**Figure 4.4: Sketch of the flow pattern of condition 1**

**Condition 2**: Three dampers secured at an angle with one damper at a different angle

The flow of the dye at the damper with the different angle began to collide with the flow at the damper next to it. It is like the flow is blocking the flow direction of the secured dampers and distorted the fluid flow upward.

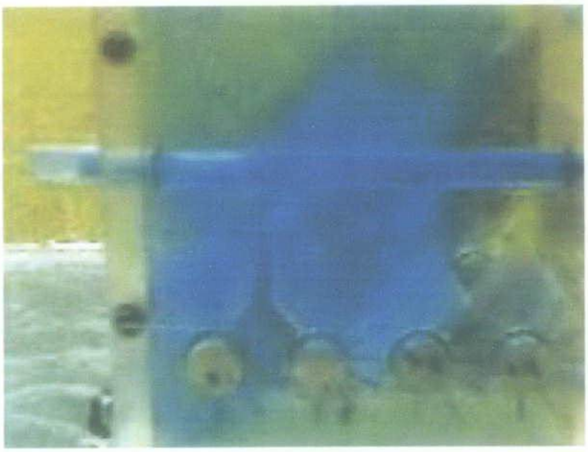

**Figure 4.5: Condition 2**

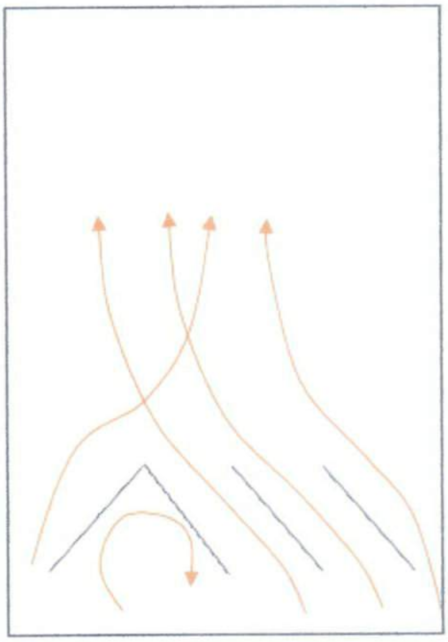

**Figure 4.6: Sketch of the flow pattern of condition 2**

**Condition 3**: Three dampers secured at an angle with one damper flapping

As the damper began to rotate itself from one side to the other, the flow seems to be swirling causing the dye to propagate in a random direction, changing its magnitude and hassling with the flow of the secured dampers. This reduces the rate of flow reaching up the top because the flow was moving from side to side rather than rising upward.

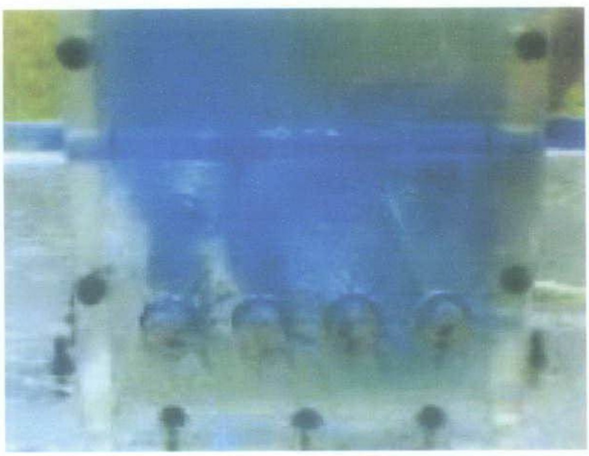

**Figure 4.7: Condition 3**

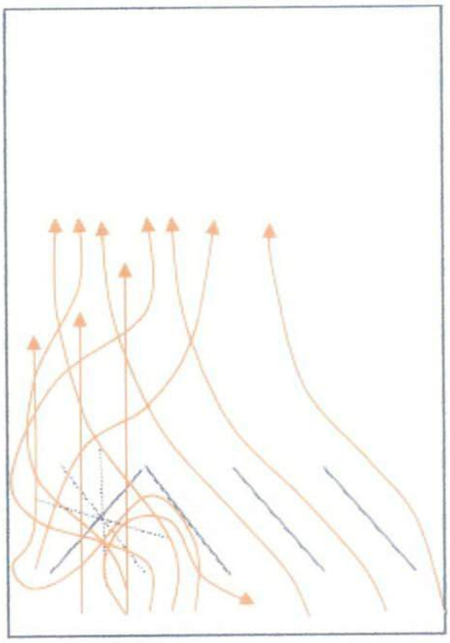

**Figure 4.8: Sketch of the flow pattern of condition 3**

### *4.2.2 Pressure reading*

**Condition 1**

The pressure at the point 1 (Refer to Figure 3.3) is the highest due to all the weight it have on top of it. The pressure reduces at gradually at point 2, 3, 4 and the lowest pressure is at point 5. The pressure remains constant at a constant flow rate and increased at the flow rate increased.

**Condition 2**

Due to the damper that is at a different angle, there will be an increased in pressure at the damper and cause the pressure at point 1 to increase as well. At the same time this will reduce the initial pressure values at points 2, 3, 4 and 5.

**Condition 3**

The flapping damper will cause the pressure to fluctuate thus giving random reading. Due to the fluctuating pressure the vibration level of the furnace's walls will also increase and this will eventually leads to failure.

### *4.2.3 Limitations*

**Dimensionless Parameters**

The Re number must seek for the similitude between the inertial and viscous forces. In real fluid flow in the furnace top, the flue gas is commonly at 200°C. In this analysis, the fluid used to model the flow is water at room temperature. When using a different fluid, such as water, with different density and dynamic viscosity values to generate a scaled down model of a prototype using air as the working fluid, there are limitations in meeting all the dimensionless parameter.

**Composition of flue gas**

The flue gas composition of the actual furnace in EPEMSB, Paka was unavailable and thus the author has used the properties of flue gas from the tables of properties obtained from Table A.4: Physical properties of flue gases to generate the Reynolds number. Ideally, the composition of flue gas should include fluids such as ethane, carbon dioxide, carbon monoxide, water, oxygen and nitrogen. The author assumed that the cracking process uses refined ethane that contains no sulphur that completely combusts to produce solely nitrogen (excess air), carbon dioxide and water molecules.

**Rotational flow**

There is a major difference between the mechanisms in which the fluid is transported in the actual furnace top and the prototype. At EPEMSB, an inductor fan is used to draw the flue gas out and hence the flow profile is as illustrated in Figure 4.9. [12]

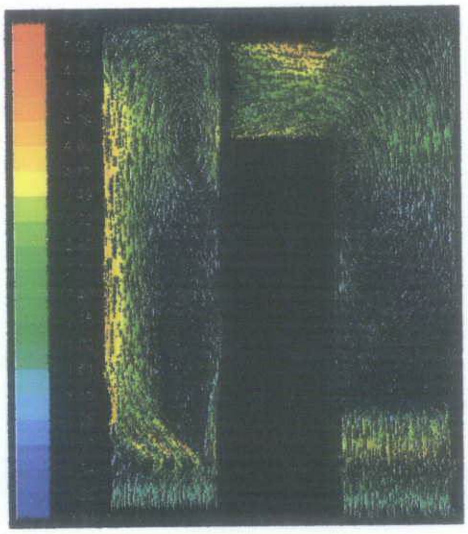

**Figure 4.9: Vectors with a fan**

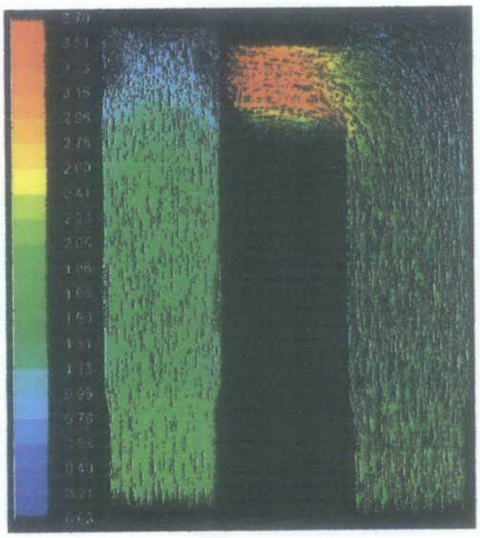

**Figure 4.10: Vectors without a fan**

However, the prototype uses a pump to force water into the rig to replace the fan as shown in Figure 4.10. [12] This was the only solution since it is more practical and economical. There are also others factors that led to the use of pump instead of fan such as;

- The exact dimensions and the speed of the inductor fan at the time of the incident were unavailable

- The driver speed of a fan cannot be controlled as easily as with a valve for a pump and such fan is unavailable in the campus.

**Pump Capacity**

Since no axial pump is available for experimental work in the campus, the author has to use a centrifugal pump which could not supply the required flow rate as calculated. The pump used can only deliver a flow rate of $0.00083 m^3/s$ and flow rate needed from the calculation is $0.0526\ m^3/s$. Despite the drawback, the results obtained have already indicated a trend of the turbulence profile especially when the dampers are flapping during testing in the experimenting stage.

**Pressure Reading**

It is very regretful that the author could not take the pressure reading with the manometer. This is because the manometer used during experimentation has a small scale while the pressure inside the prototype is quite high. As soon as water filled the prototype, the pressure at each tapping has already gone of scale.

# CHAPTER 5
# RECOMMENDATIONS AND CONCLUSIONS

## 5.1 RECOMMENDATIONS

There are two recommendations that will be suggested in this section regarding the enhancement of the actual furnace top and the prototype itself.

### *5.1.1 Recommendations for the furnace top*

Some modifications should be done to the furnace top. This include of redesigning the shape of the furnace. By eliminating corners and sharp bends inside the furnace, the flow will be more stabilized as it will decrease the pressure induced inside the furnace.(Figure 5.1)

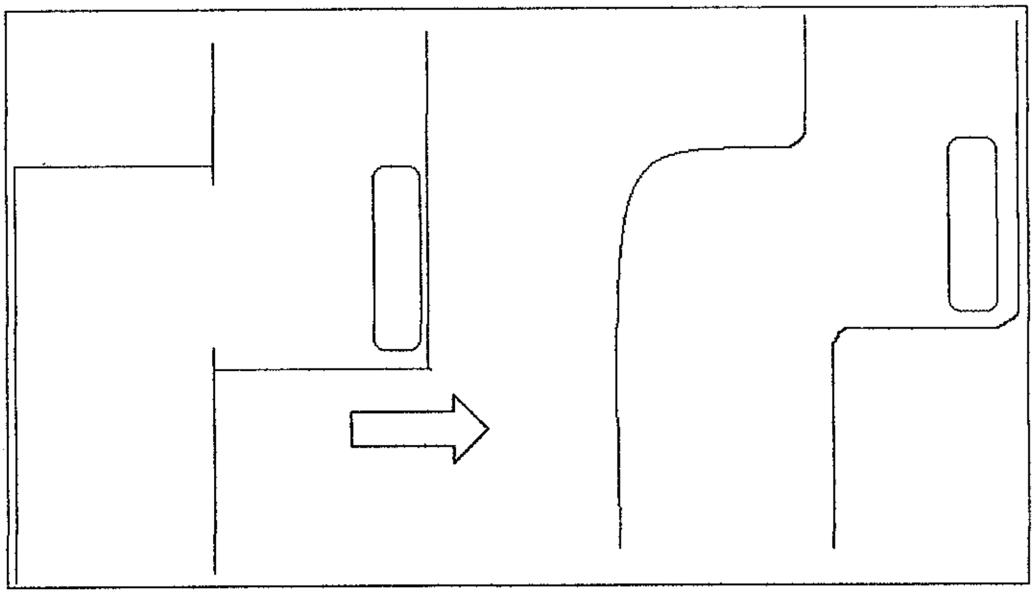

**Figure 5.1 Design modification of the furnace top**

An electronic locking mechanism should be installed to the dampers. If any of the cotter pin breaks, the dampers will be lock at their respective place and the flow will remain steady. An electronic pressure sensor will also help as it can detect any pressure fluctuation inside the furnace to prevent vibrations from the early stage. However the sensor must withstand high pressure and temperature to be inside the furnace. Preventive maintenance should take place regularly by doing inspection to maintain the performance of the furnace.

### *5.1.2 Recommendations for the test rig/prototype*

- Damper flappers are controlled using gears to enable a full span of rotation from completely open to completely shut.
- Use an inductor fan in place of a pump to create the rotational flow effect.
- A proper square duct which enables the flow to be fully developed before entering the test section.
- Use a suitable pressure measuring device that can get an accurate reading. The author has suggested a custom manometer to take the pressure reading.
- The prototype should be sealed completely to avoid pressure loss due to leakage.

## 5.2 Conclusion

Assumptions made during calculations may have resulted in deviation from the actual result. However the assumptions are essential because of unknown variables or insufficient information during designing the prototype of the furnace.

The first objective of this project is to modify the existing prototype to match the actual furnace at a scale of 1/10. However, by referring to the actual drawing, it seems that a new prototype has to design and fabricated to match the actual furnace. The prototype was fabricated and the first objective was fulfilled. The prototype was design by using the AutoCAD software and essential values for experimentation were achieved by referring to the dimensional parameters for similarities. The dimensionless parameter matched (Re) allowed an analysis of the effects of velocity differences on the force exerted on the test rig components. The dimensionless parameter (Cp) could not be exactly matched since the pressure at the inlet and outlet of the furnace are unknown.

Secondly, during experimentation the flow visualization was quite a success but the author failed to get a precise pressure reading on the pressure taps installed. A custom made manometer was suggested in the recommendation which consist of a larger scale in order to acquire the pressure reading.

Through the flow visualization, induced turbulence can be detected when the flapper is rotating loosely. The amount of turbulence inside the furnace is greater when;

a) The flow rate of fluid into the furnace increases

b) The loose damper flapper rotates at a higher frequency.

c) The angles of the damper flappers are more deviated with respect to the vertical axis.

Apart from the loose damper, the geometry of the furnace is also assumed to cause vibration due to induce pressure that causes the flow to disrupt. By eliminating sharp corners and bend inside the furnace, the flow can be more stabilized since it can reduce the pressure induction within the flow.

# REFERENCES

[1] Yunus A. Cengel and John M. Cimbala ,FLUID MECHANICS, Fundamentals and Applications, INTERNATIONAL EDITION, McGRAW.HILL (2006)

[2] Micheal J. Moran and Howard N. Shapiro,*Fundamentals of Engineering Thermodynamics* 5$^{TH}$ Edition, John Wiley & Sons, Inc. (2004)

[3] Jasbir Singh. *Heat Transfer Fluids and Systems for Process and Energy Applications.*

[4] Tom Bitzer, *Honeycomb Technology: Materials, Design, Manufacturing, Applications and Testing,* 3$^{rd}$ Edition, Chapman and Hall (2007)

[5] Engineering Toolbox <http://www.engineeringtoolbox.com>

[6] http://www.cocreate.com

[7] Engineering Fundamentals http://www.efunda.com

[8] Thanavathy Patma Nesan *Experimental Modeling of the Flow in the Furnace Top of an Ethane Cracker at EPEMSB Paka,* Final Year Thesis, Universiti Teknologi Petronas (December 2007)

[9] Alif Syazwan bin Mohd Nasir. *Experimental Modeling of the Flow in the Furnace Top of an Ethane Cracker at EPEMSB Paka,* Final Year Thesis, Universiti Teknologi Petronas (June 2008)

[10] Non-catalytic pyrolysis of ethane to ethylene in the presence of $CO_2$ with or without limited $O_2$ by Vasant R Chaudhary

[11] Lab Manual, Thermofluid Lab EMB3052

[12] Tomi Tulin Entaban *Simulation of a Fluid Flow in a Furnace Top in EPEMSB, Paka,* Final Year Thesis, Universiti Teknologi Petronas (December 2007)

# APPENDIX A

# TABLES

### Table A.1: Primary dimensions and their symbols, SI, English and BG units

| Primary Dimension | Symbol | SI unit | BG unit | English unit |
|---|---|---|---|---|
| mass | m (sometimes M) | kg (kilogram) | slug | lbm (pound-mass) |
| length | L (sometimes l) | m (meter) | ft (foot) | ft (foot) |
| time | t (sometimes T) | s (second) | s (second) | s (second) |
| temperature | T (sometimes θ ) | K (Kelvin) | °R (degree Rankine) | R (Rankine) |
| electric current | I (sometimes i) | A (ampere) | A (ampere) | A (ampere) |
| amount of light (luminous intensity) | C (sometimes I) | c (candela) | c (candela) | c (candela) |
| amount of matter | n or N (sometimes μ) | mol (mole) | mol (mole) | mol (mole) |

### Table A.2: Common quantity with their primary dimensions

| | | Dimensions | |
|---|---|---|---|
| Quantity | Symbol | $MLT\Theta$ | $FLT\Theta$ |
| Length | $L$ | $L$ | $L$ |
| Area | $A$ | $L^2$ | $L^2$ |
| Volume | $V$ | $L^3$ | $L^3$ |
| Velocity | $V$ | $LT^{-1}$ | $LT^{-1}$ |
| Acceleration | $dV/dt$ | $LT^{-2}$ | $LT^{-2}$ |
| Speed of sound | $a$ | $LT^{-1}$ | $LT^{-1}$ |
| Volume flow | $Q$ | $L^3T^{-1}$ | $L^3T^{-1}$ |
| Mass flow | $\dot{m}$ | $MT^{-1}$ | $FTL^{-1}$ |
| Pressure, stress | $p, \sigma$ | $ML^{-1}T^{-2}$ | $FL^{-2}$ |
| Strain rate | $\dot{\varepsilon}$ | $T^{-1}$ | $T^{-1}$ |
| Angle | $\theta$ | None | None |
| Angular velocity | $\omega$ | $T^{-1}$ | $T^{-1}$ |
| Viscosity | $\mu$ | $ML^{-1}T^{-1}$ | $FTL^{-2}$ |
| Kinematic viscosity | $\nu$ | $L^2T^{-1}$ | $L^2T^{-1}$ |
| Surface tension | $Y$ | $MT^{-2}$ | $FL^{-1}$ |
| Force | $F$ | $MLT^{-2}$ | $F$ |
| Moment, torque | $M$ | $ML^2T^{-2}$ | $FL$ |
| Power | $P$ | $ML^2T^{-3}$ | $FLT^{-1}$ |
| Work, energy | $W, E$ | $ML^2T^{-2}$ | $FL$ |
| Density | $\rho$ | $ML^{-3}$ | $FT^2L^{-4}$ |
| Temperature | $T$ | $\Theta$ | $\Theta$ |
| Specific heat | $c_p, c_v$ | $L^2T^{-2}\Theta^{-1}$ | $L^2T^{-2}\Theta^{-1}$ |
| Specific weight | $\gamma$ | $ML^{-2}T^{-2}$ | $FL^{-3}$ |
| Thermal conductivity | $k$ | $MLT^{-3}\Theta^{-1}$ | $FT^{-1}\Theta^{-1}$ |
| Expansion coefficient | $\beta$ | $\Theta^{-1}$ | $\Theta^{-1}$ |

## Table A.3: Some common established nondimensional parameter or Πs encountered in fluid mechanics and heat transfer

| Parameter | Definition | Qualitative ratio of effects | Importance |
|---|---|---|---|
| Reynolds number | $\mathrm{Re} = \frac{\rho U L}{\mu}$ | $\frac{\text{Inertia}}{\text{Viscosity}}$ | Always |
| Mach number | $\mathrm{Ma} = \frac{U}{a}$ | $\frac{\text{Flow speed}}{\text{Sound speed}}$ | Compressible flow |
| Froude number | $\mathrm{Fr} = \frac{U^2}{gL}$ | $\frac{\text{Inertia}}{\text{Gravity}}$ | Free-surface flow |
| Weber number | $\mathrm{We} = \frac{\rho U^2 L}{\Upsilon}$ | $\frac{\text{Inertia}}{\text{Surface tension}}$ | Free-surface flow |
| Cavitation number (Euler number) | $\mathrm{Ca} = \frac{p - p_v}{\rho U^2}$ | $\frac{\text{Pressure}}{\text{Inertia}}$ | Cavitation |
| Prandtl number | $\mathrm{Pr} = \frac{\mu c_p}{k}$ | $\frac{\text{Dissipation}}{\text{Conduction}}$ | Heat convection |
| Eckert number | $\mathrm{Ec} = \frac{U^2}{c_p T_0}$ | $\frac{\text{Kinetic energy}}{\text{Enthalpy}}$ | Dissipation |
| Specific-heat ratio | $k = \frac{c_p}{c_v}$ | $\frac{\text{Enthalpy}}{\text{Internal energy}}$ | Compressible flow |
| Strouhal number | $\mathrm{St} = \frac{\omega L}{U}$ | $\frac{\text{Oscillation}}{\text{Mean speed}}$ | Oscillating flow |
| Roughness ratio | $\frac{\epsilon}{L}$ | $\frac{\text{Wall roughness}}{\text{Body length}}$ | Turbulent, rough walls |
| Grashof number | $\mathrm{Gr} = \frac{\beta \Delta T g L^3 \rho^2}{\mu^2}$ | $\frac{\text{Buoyancy}}{\text{Viscosity}}$ | Natural convection |
| Temperature ratio | $\frac{T_w}{T_0}$ | $\frac{\text{Wall temperature}}{\text{Stream temperature}}$ | Heat transfer |
| Pressure coefficient | $C_p = \frac{p - p_\infty}{\frac{1}{2}\rho U^2}$ | $\frac{\text{Static pressure}}{\text{Dynamic pressure}}$ | Aerodynamics, hydrodynamics |
| Lift coefficient | $C_L = \frac{L}{\frac{1}{2}\rho U^2 A}$ | $\frac{\text{Lift force}}{\text{Dynamic force}}$ | Aerodynamics, hydrodynamics |
| Drag coefficient | $C_D = \frac{D}{\frac{1}{2}\rho U^2 A}$ | $\frac{\text{Drag force}}{\text{Dynamic force}}$ | Aerodynamics, hydrodynamics |

**Table A.4: Physical properties of flue gases**

## Physical propetries of flue gases

Availabile tables: dry air gases flue gases water steam

This table is for flue gases. It gives values of some physical properties in relation to the temperature of gases.

It is for following chemical composition:

- carbon dioxide $CO_2$ - 13%
- water vapour $H_2O$ - 11%
- nitrogen $N_2$ - 76%

| t | rho | cp | mi*10^6 | ni*10^6 |
|---|---|---|---|---|
| [°C] | [kg/m³] | [kJ/kgK] | [Pas] | [m²/s] |
| 0 | 1.295 | 1.042 | 15.8 | 12.2 |
| 100 | 0.95 | 1.068 | 20.4 | 21.54 |
| 200 | 0.748 | 1.097 | 24.5 | 32.8 |
| 300 | 0.617 | 1.122 | 28.2 | 45.81 |
| 400 | 0.525 | 1.151 | 31.7 | 60.38 |
| 500 | 0.457 | 1.185 | 34.8 | 76.3 |
| 600 | 0.405 | 1.214 | 37.9 | 93.61 |
| 700 | 0.363 | 1.239 | 40.7 | 112.1 |
| 800 | 0.33 | 1.264 | 43.4 | 131.8 |
| 900 | 0.301 | 1.29 | 45.9 | 152.5 |
| 1000 | 0.275 | 1.306 | 48.4 | 174.3 |
| 1100 | 0.257 | 1.323 | 50.7 | 197.1 |
| 1200 | 0.24 | 1.34 | 53 | 221 |

where is for flue gas:

- t - temperature
- rho - density
- $c_p$ - specific heat
- mi - dynamic viscosity
- ni - kinematic viscosity

# APPENDIX B

## FIGURES

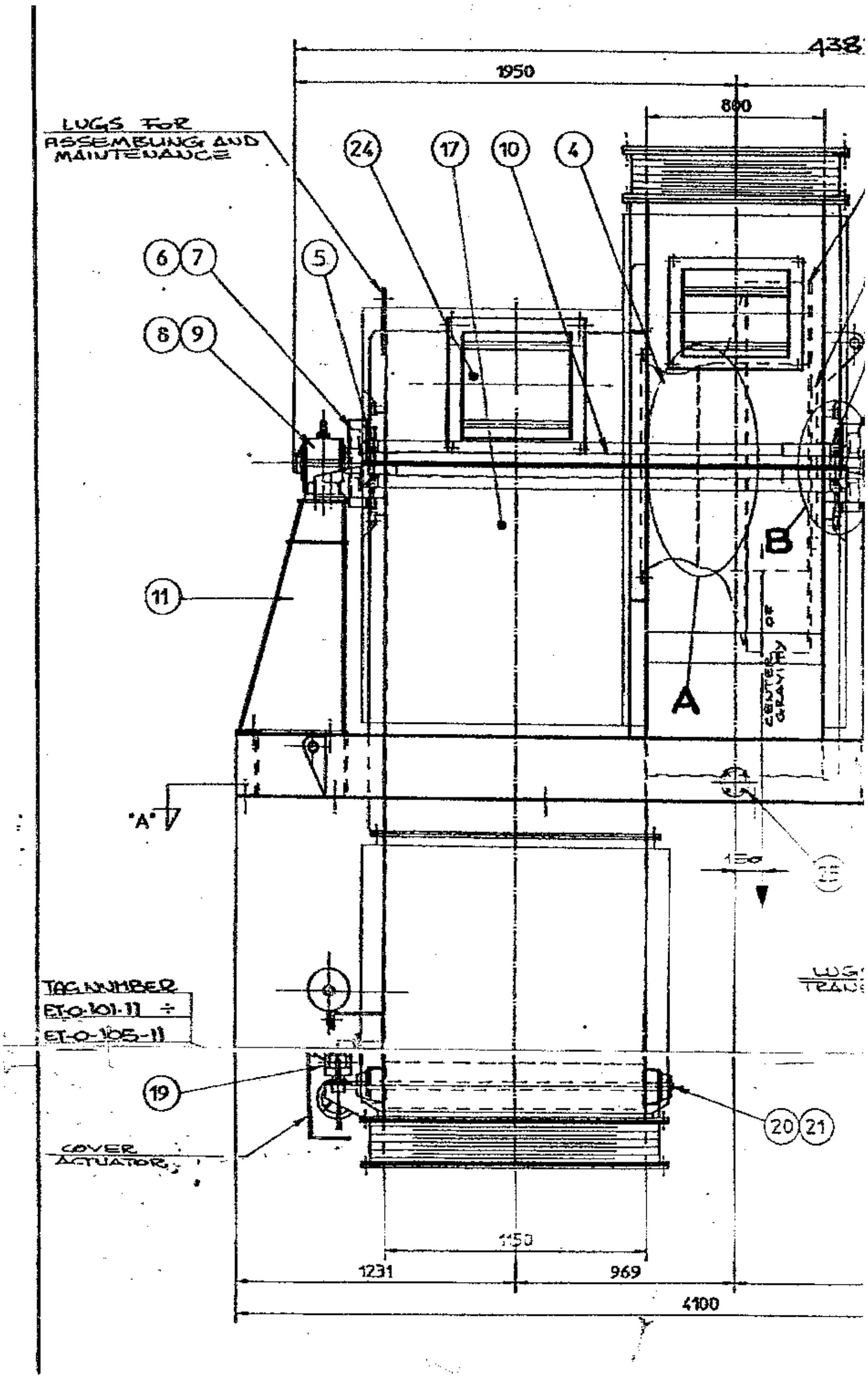


**Figure B.2: Side view drawing of the actual furnace at EPEMSB Paka.**

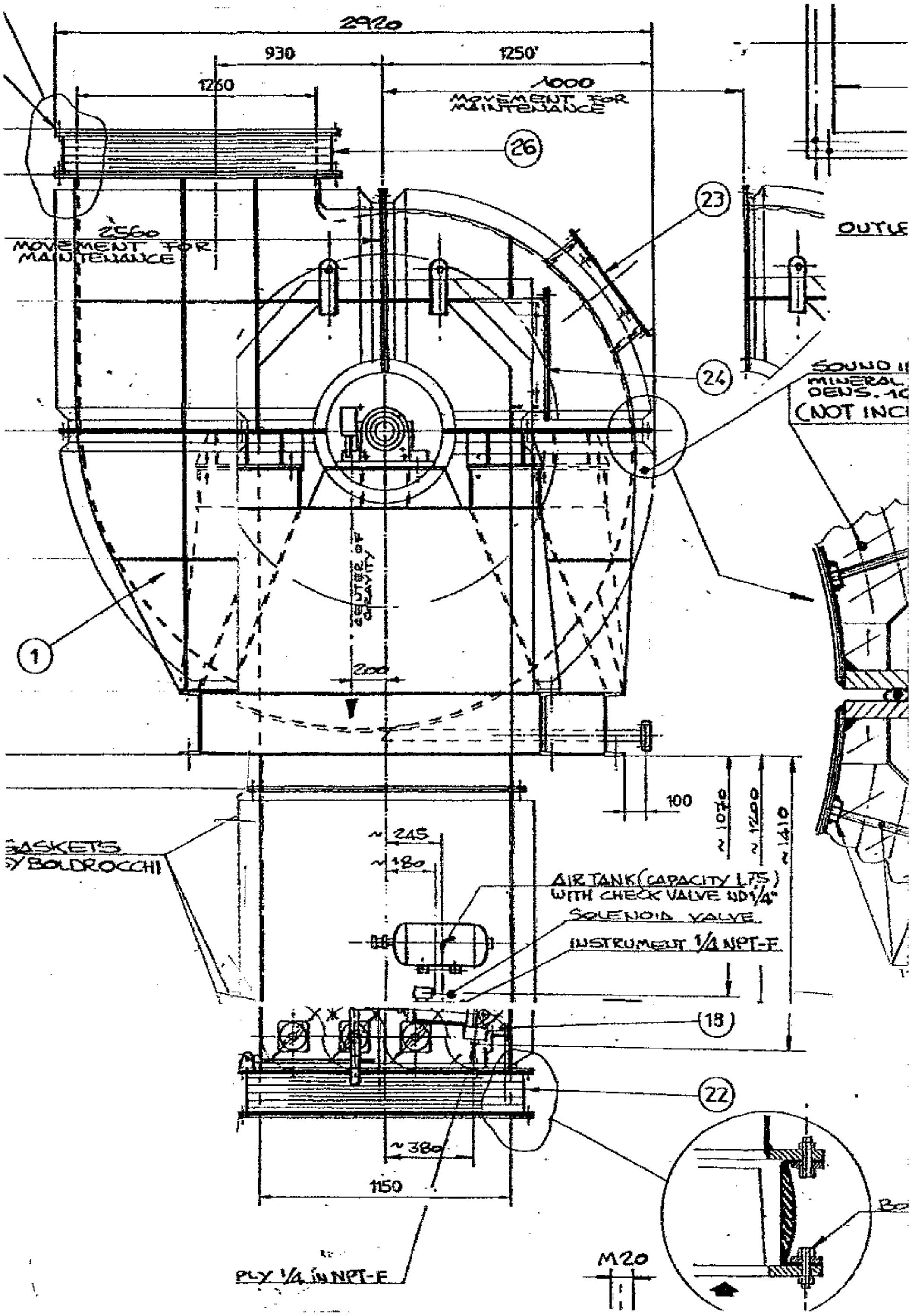


Figure B.3: Front view drawing of the actual furnace at EPEMSB Paka.

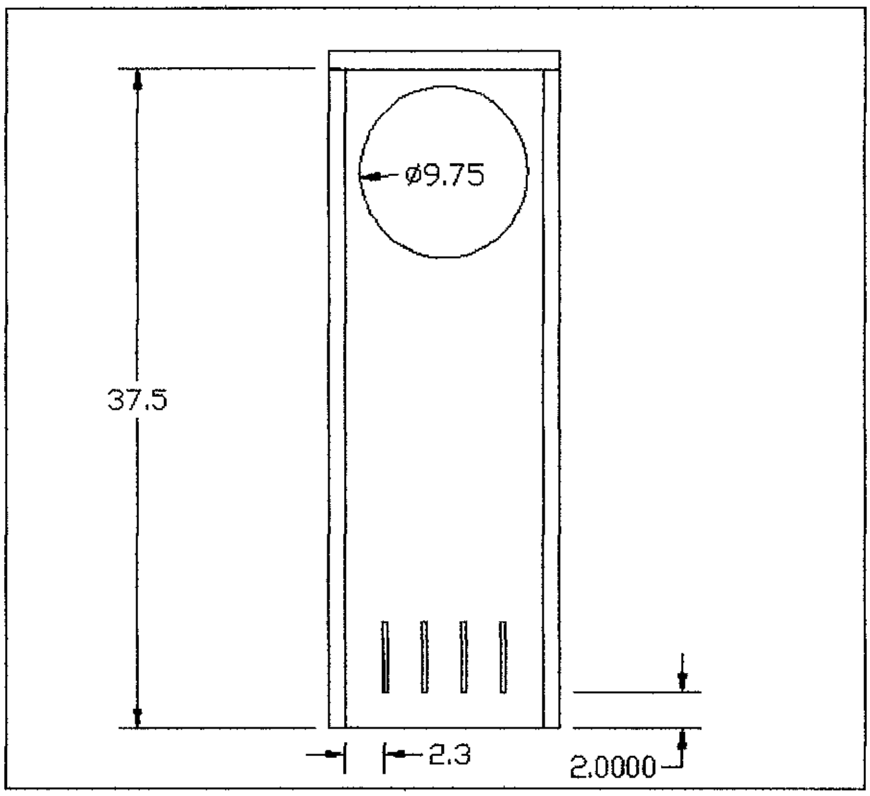


Figure B.4: Front view of the prototype (Dimensions are in millimeter, mm)

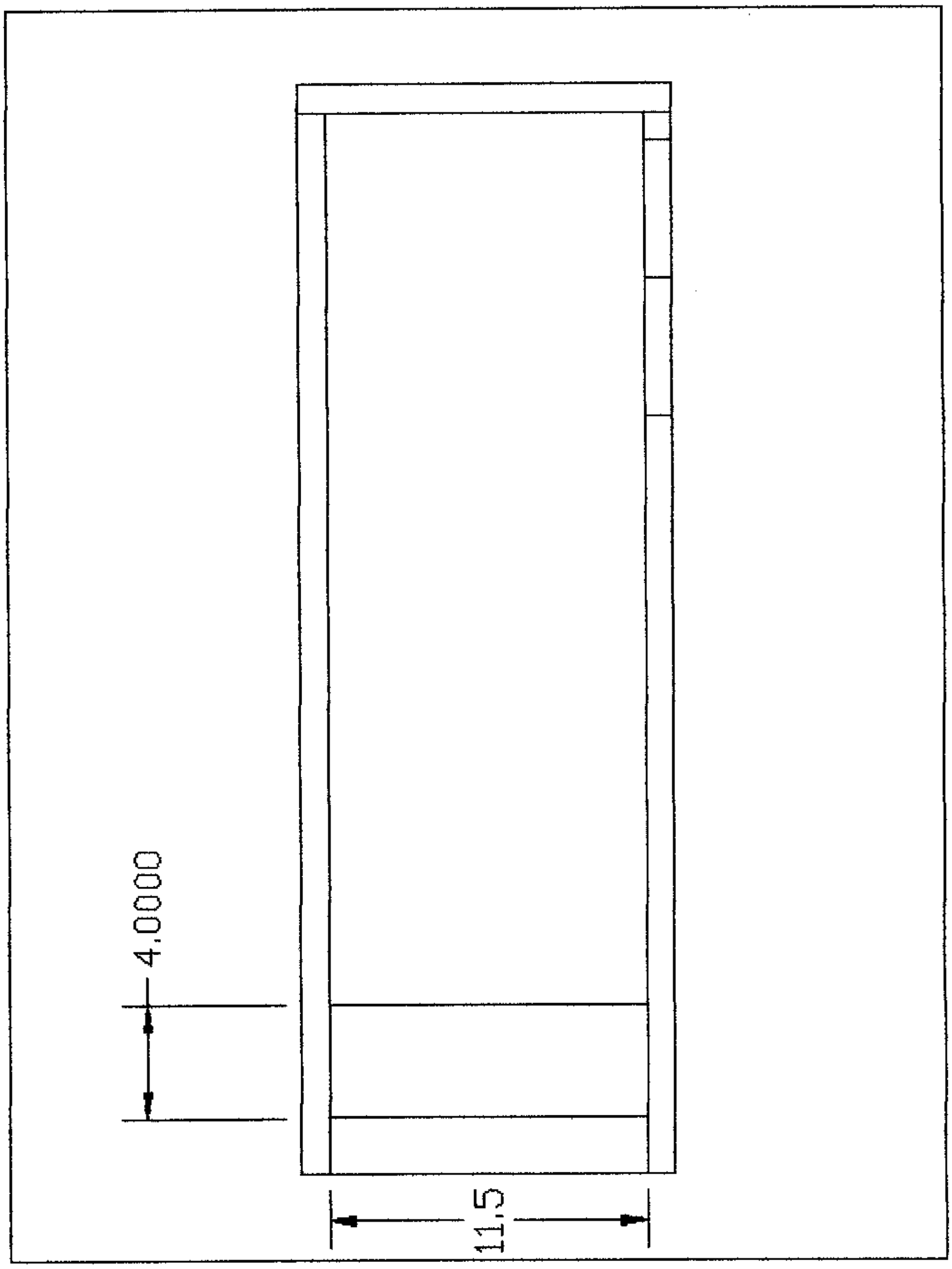


**Figure B.5: Side view of the prototype**

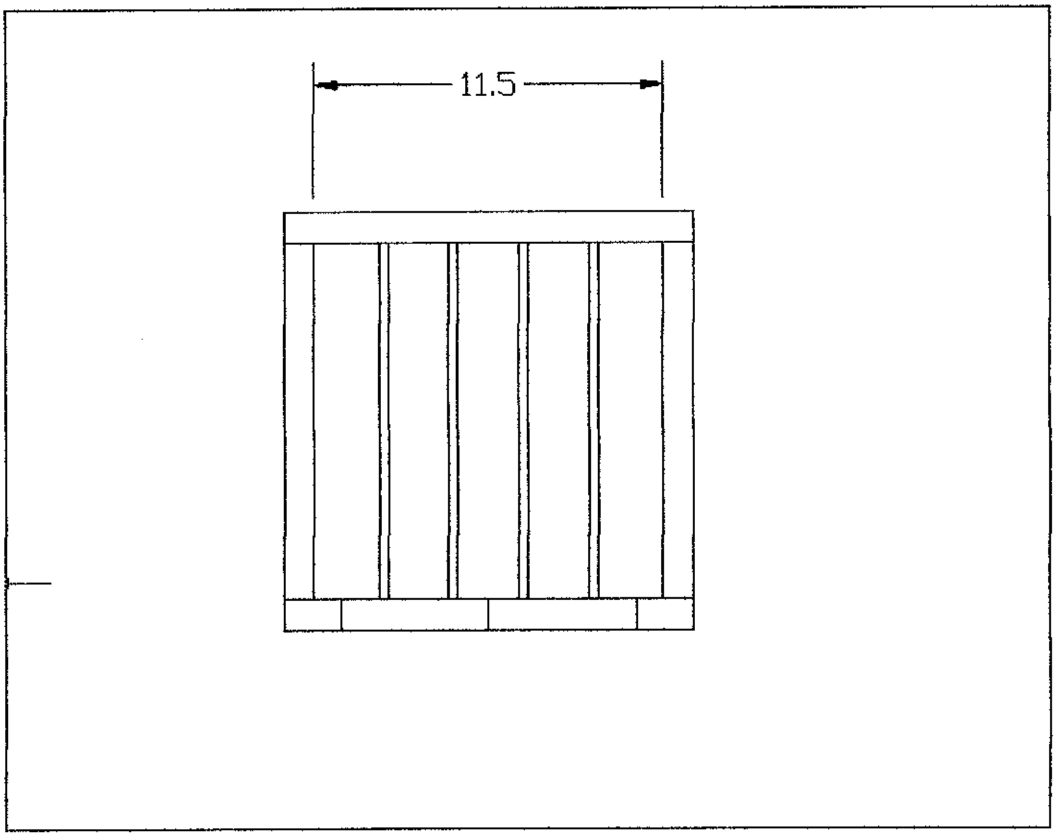


**Figure B.6: Plan view of the prototype**

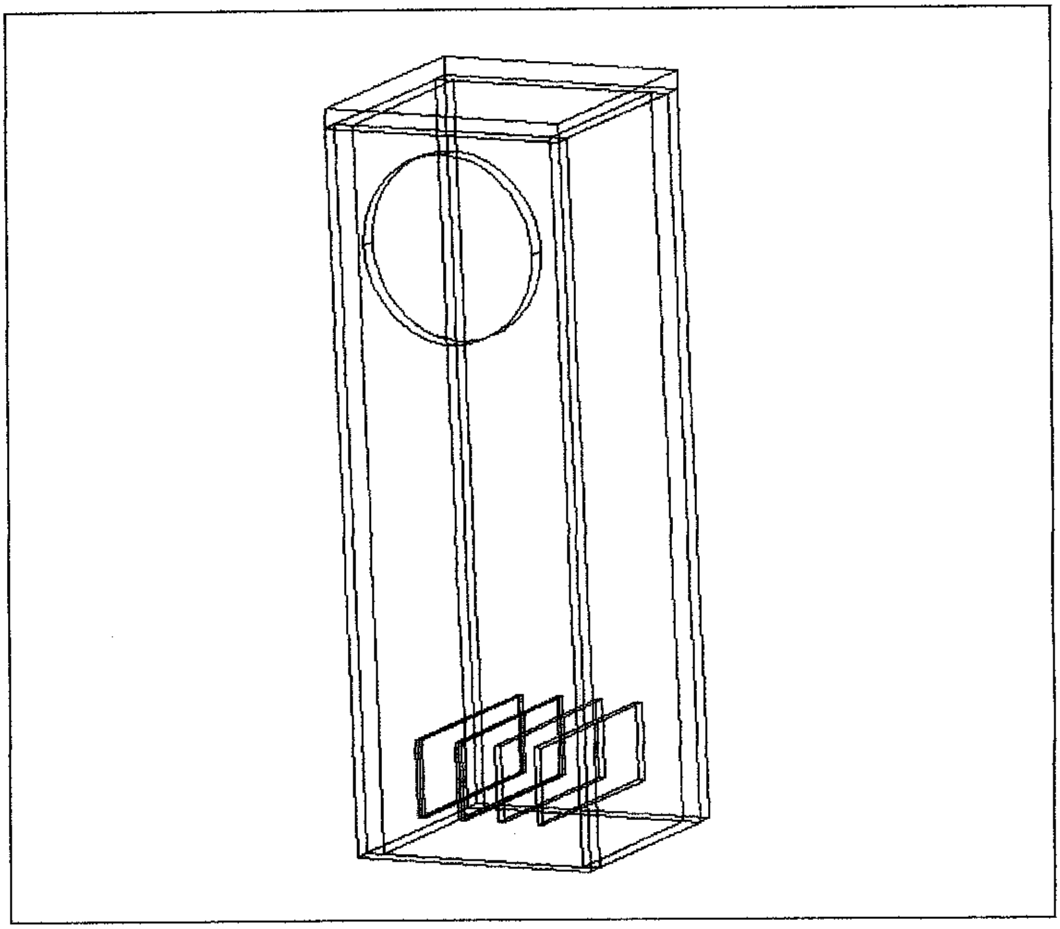

**Figure B.7: Oblique drawing (3D) of the prototype**

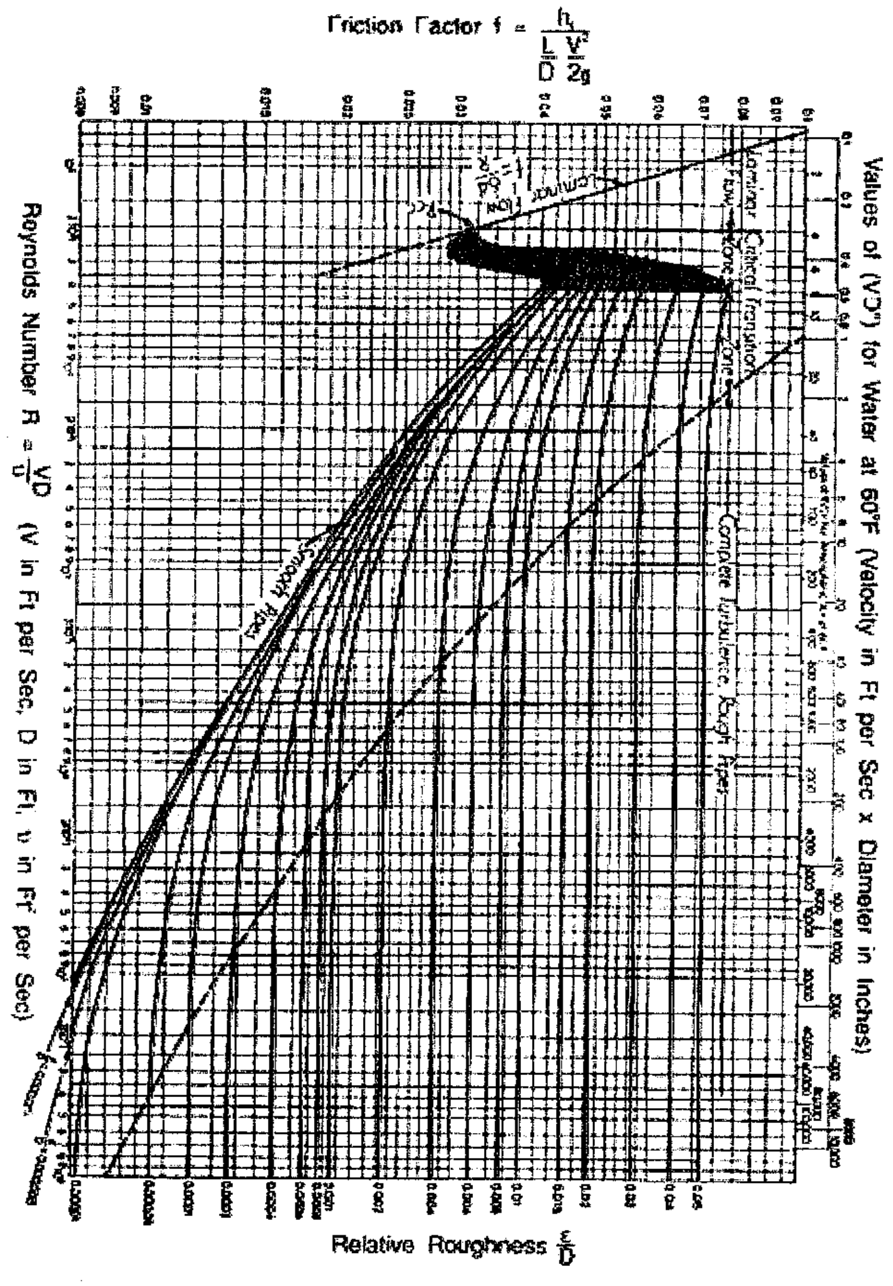


**Figure B.8: Moody Chart**