UNIVERSITI MALAYSIA PAHANG

BORANG PENGESAHAN STATUS TESIS

JUDUL: ANALYZE THE SPRINKLER SYSTEM IN UMP LECTURE ROOM USING CFD (COMPUTATIONAL FLUID DYNAMICS) ANALYSIS

SESI PENGAJIAN: 2007/2008

Saya **MUHAMAD KHUMAINI BIN KUZAIMAN**

mengaku membenarkan tesis (PSM/Sarjana/Doktor Falsafah)* ini disimpan di Knowledge Management Centre dengan syarat-syarat kegunaan seperti berikut:

1. Tesis adalah hakmilik Universiti Malaysia PAHANG
2. Knowledge Management Centre dibenarkan membuat salinan untuk tujuan pengajian sahaja.
3. Perpustakaan dibenarkan membuat salinan tesis ini sebagai bahan pertukaran antara institusi pengajian tinggi.
4. **Sila tandakan (✓)

[ ]	SULIT	(Mengandungi maklumat yang berdarjah keselamatan atau kepentingan Malaysia seperti yang termaktub di dalam AKTA RAHSIA RASMI 1972)
[ ]	TERHAD	(Mengandungi maklumat TERHAD yang telah ditentukan oleh Organisasi/badan dimana penyelidikan dijalankan)
[✓]	TIDAK TERHAD	

Disahkan oleh

(TANDATANGAN PENULIS) (TANDATANGAN PENYELIA)

Alamat Tetap: 4-B, KAMPUNG KELUANG
22200 BESUT
TERENGGANU

Nama Penyelia: DEVARAJAN A/L RAMASAMY

Tarikh: 8/5/2008

Tarikh: 8/5/2008

CATATAN: * Potong yang tidak berkenaan

** Jika tesis ini SULIT atau TERHAD, sila lampirkan surat daripada pihak berkuasa/organisasi berkenaan dengan menyatakan sekali sebab dan tempoh tesis ini perlu dikelaskan sebagai SULIT atau TERHAD

υ Tesis dimaksudkan sebagai tesis bagi Ijazah Doktor Falsafah dan Sarjana secara penyelidikan, atau disertasi bagi pengajian secara kerja kursus dan penyelidikan, atau Laporan Projek Sarjana Muda (PSM)

ANALYZE THE SPRINKLER SYSTEM IN UMP LECTURE ROOM USING CFD (COMPUTATIONAL FLUID DYNAMICS) ANALYSIS

MUHAMAD KHUMAINI BIN KUZAIMAN

A report submitted in partial fulfillment
of the requirements for the award of the degree of
Bachelor of Mechanical Engineering

Faculty of Mechanical Engineering
Universiti Malaysia Pahang

MAY 2008

SUPERVISOR DECLARATION

We hereby declare that we have read this thesis and in our opinion this thesis is sufficient in terms of scope and quality for the award of the degree of Mechanical Engineering.

Signature :

Supervisor : MR DEVARAJAN A/L RAMASAMY

Date : 8/5/2008

STUDENT DECLARATION

"All the trademark and copyrights use herein are property of their respective owner. References of information from other sources are quoted accordingly; otherwise the information presented in this report is solely work of the author."

Signature : ____________________

Author :MUHAMAD KHUMAINI BIN KUZAIMAN

Date : 8/05/2008

DEDICATION

To my beloved mother and father

NORIZAM BT HAMZAH

KUZAIMAN B ABDULLAH

ACKNOWLEDGEMENT

Alhamdulillah, the highest thanks to God because only with His Willingness, it is possible to complete this final year project in time.

I would like to express my gratitude to my dedicated supervisor, Mr Devarajan A/L Ramasamy for guiding this project with clarity and that priceless gift of getting things done by sharing his valuable ideas as well as his knowledge. Thanks also to UMP Mechanical Engineer Mr Izwan B Ismail for borrowing me Anemometer for this project experiment..

I also like to thank to all UMP lecturers and technicians whom have helped directly or indirectly in whatsoever manner thus making this project a successful. Not forgotten are my best colleagues for their openhandedly and kindly guided, assisted, and supported and encouraged me to make this project successful. My heartfelt thank to my dearest family which always support me for this project. Their blessing gave me the high-spirit and strength to face any problem occurred and to overcome them rightly.

The great cooperation, kindheartedness and readiness to share worth experiences that have been shown by them will be always appreciated and treasured by me. Once again, thank you very much.

ABSTRACT

An automatic sprinkler system is one of the most effective methods of controlling or suppressing the fire. Sprinklers are generally located at the ceiling level of the level of a building. They are equipped with a fusible link that melts when the heat given off by a fire heats the sprinkler. Currently there is no sprinklers system that installed in UMP (Universiti Malaysia Pahang) lecture room. In this project by using *CFD (Computational Fluid Dynamics) a simulation of fire occurred is created to prove* the effect of sprinkler in one of the lecture room (Block Y-BK8). For this simulation, assume that the fire happen due to shock circuit at personal computer. Now from CFD, simulation required finding out boundary condition in that room like power output of lamp and PC. Also air flow rate by air conditioner and sprinkler. From the result of simulation, the sprinkler system is effective if the fire occurred in the lecture room. As a conclusion, a sprinkler system can be installed as a precaution at the lecture room.

ABSTRAK

Sistem sprinkler yang automatik adalah kaedah yang paling berkesan untuk memadam dan mengawal kebakaran. Sprinkler secara umumnya terletak kat atas siling bangunan. Ia dilengkapi dengan fius yang akan cair bila terkena haba. Pada masa ini, tiada sistem sprinkler yang di pasang di bilik kuliah kat Universiti Malaysia Pahang (UMP).Dalam projek ini, dengan mengunakan CFD satu simulasi kebakaran maya dicipta *untuk membuktikan keberkesanan sprinkler dalam satu bilik kuliah* (Blok Y-BK8). *Untuk* simulasi ini, diandaikan kebakaran terjadi kerana litar pintas dekat PC. Untuk menjayakan simulasi ini, bacaan dari keadaan sekeliling seperti haba dikeluarkan oleh PC dan lampu diperlukan. Selain itu halaju udara dikeluarkan oleh penyaman udara dan sprinkler. Daripada keputusan simulasi, dapat dibuktikan keberkesanan sprinkler dalam menangani kebakaran yang berlaku di bilik kuliah. Sebagai kesimpulan, sistem sprinkler boleh dipasang sebagai langkah berjaga-jaga di bilik kuliah.

TABLE OF CONTENTS

LIST OF TABLES

LIST OF FIGURES

LIST OF NOMENCLATURE

Roman Symbol

V	Velocity
ν	Viscosity
Re	Reynolds Number
ρ	Density
T	Temperature
L	Length
D	Diameter

LIST OF APPENDICES

CHAPTER 1

INTRODUCTION

1.1 Project background

Sprinkler system is used to contain fire and in many cases put it out, even before the fire department arrives. Currently, in Universiti Malaysia Pahang (UMP) there is no sprinkler system installed in lecture rooms. There is possibility that the room may have a fire due to short circuit at electronic appliances such as computer in the room. So, to prevent hazard to the room, sprinkler system is essentially needed. Usually the sprinkler system is installed following the rule set by fire department.

1.2 Objective

Using CFD to analyze the effect of sprinkler system in the lecture room at UMP (Block Y-DK8)

1.3 Scope of Study

The scopes of this project are divided into 3 mains part:

a) Predict fire propagation inside the room
b) Understand sprinkler system
c) Using CFD to analyze fire in the room

1.4 Flow Chart

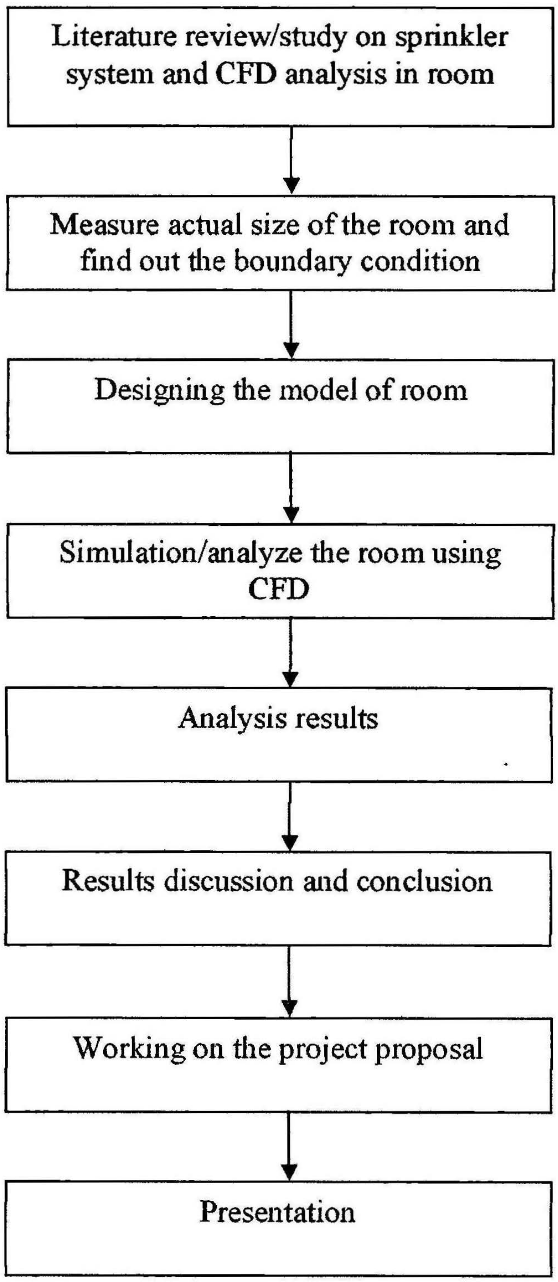


Figure 1.1: Flow chart

CHAPTER 2

LITERATURE STUDY

2.1 Fire

Fire is essentially a chemical reaction known as combustion. It can spread in minutes and kill in second. Knowing the characteristics of fire and understanding how can help Architects, Engineers and other professional to formulate strategies on life safety and property protection in building designs.

Three factors are needed to start a fire are FUEL, OXYGEN AND HEAT. This is known as the "Fire Triangle". If this chemical reaction is allowed spread unchecked, a very small fire will quickly develop into an inferno and involve the entire building.

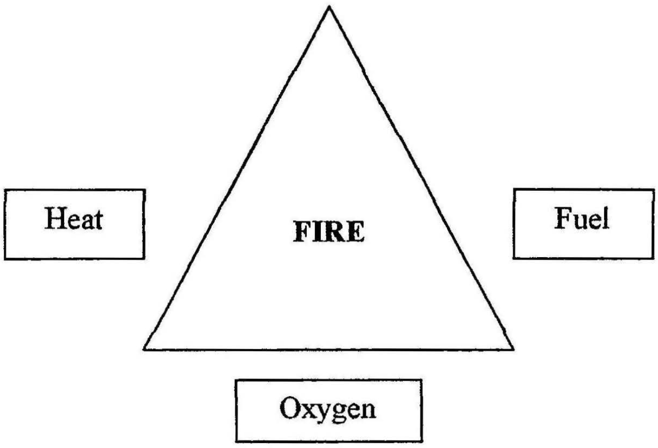


Figure 2.1: Fire Triangle (Prof Datuk Dr Soh Chai Hock, 1998).

2.1.1 Fire Growth Stages

2.1.1.1 IGNITION

Combustion can be very fast, as in a gas explosion or it can be slow smoudering process.

2.1.1.2 GROWTH

A fire can once started can grow rapidly as it creates the conditions for its own growth. In an enclosed compartment, a critical stage may be reached where all the combustion materials are heated to flammable concentrations of gases and the fire suddenly flashes through out the whole compartment. This is known as a "Flashover".

2.1.1.2 DEVELOPMENT

The fire passes through a development stage after the initial rapid growth. During this stage the fire temperature increases more slowly. However the fire continues to spread into other areas, which then in turn continues the process of rapid initial growth.

2.1.1.3 DECAY

In the final stage, the fire will burn itself out due to lack of fuel or oxygen. In an enclosed compartment, a fire may smolder for a long period. A sudden rush of oxygen, example breaking a window, can reignite the fire with explosive violence.

2.1.1.4 FIRE LOAD

The total amount of combustible materials available to fire. Certain combustible materials released more heat than others when they burn and so contribute more to the fire load. For example 1 kg of kerosene will release 46050 kJ of heat where as the same weight of paper releases about 16900 kJ.

2.1.1.2 DEVELOPMENT

The fire passes through a development stage after the initial rapid growth. During this stage the fire temperature increases more slowly. However the fire continues to spread into other areas, which then in turn continues the process of rapid initial growth.

2.1.1.3 DECAY

In the final stage, the fire will burn itself out due to lack of fuel or oxygen. In an enclosed compartment, a fire may smolder for a long period. A sudden rush of oxygen, example breaking a window, can reignite the fire with explosive violence.

2.1.1.4 FIRE LOAD

The total amount of combustible materials available to fire. Certain combustible materials released more heat than others when they burn and so contribute more to the fire load. For example 1 kg of kerosene will release 46050 kJ of heat where as the same weight of paper releases about 16900 kJ.

2.1.2 How Fire Spread

Through natural laws, heat and smoke will travel from hotter to cooler areas by any of the tree methods.

2.1.2.1 Convection

More than 75 percent of the combustion products of a fire example smoke, burning brands, toxic gases are dissipated in rising convection currents of hot gases at temperatures of 800-1000 degree centigrade which heat anything in their path. It will create a "mushroom effect", when the convection current is blocked example by underside of floor or ceiling. It can also smoke log escape routes and prevent means of escape (Prof Datuk Dr Soh Chai Hock, 1998).

2.1.2.2 Radiation

Radiant heat is transmitted to buildings or material not shielded from fire –it is the transfer of heat energy as electromagnetic waves. Radiation passes through normal glass window easily, and building with many or large windows are more likely to spread fire to other building.

2.1.2.3 Conduction

The movement of heat through materials example metals is better conductors of heat than stones. Conducted heat can travel through partitions, floor, ceilings, and walls, to adjacent rooms, especially through metal piping, metal frames and joists. Combustible materials or internal linings of adjacent rooms can be heated to their ignition temperature by conducted heat.

2.2 Sprinkler System

An automatic sprinkler system is intended to detect, control and extinguish a fire, and warn the occupants of the occurrence of fire. The installation comprises fire pumps, water storage tanks, control valve sets, sprinkler heads, flow switches, pressure switches, pipe work and valves. The system operates automatically without human intervention.

Sprinkler is equipped with high water pressure to put out fire instantly like fireman hose. It is made with specific pipe called Chlorinated Polyvinyl Chloride or CPVC and discharged about 25 gallons water per minute (around 113.65 liters per minute). The sprinkler is equipped with a fusible link that melts when the heat given off by a fire and then the water is spray out. For example:

2.2.1 Automatic Sprinklers Model HC Bulb Spray Series

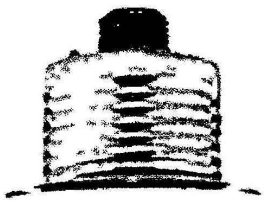


Figure 2.2: Sprinkler

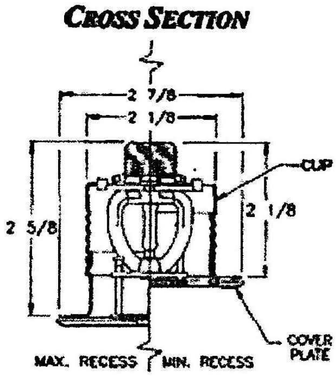


Figure 2.3: Cross Section of Sprinkler

2.2.1.1 Description and Operation

The Globe Model HC Adjustable Concealed Sprinkler is an independent sprinkler designed for use with concealed piping systems for those areas where an attractive appearance is of primary importance. The Model HC is an assembly provided with a cover plate which completely hides the sprinkler head. After installation all that is visible is the small flat cover which can be factory painted any color you wish. *Incorporated in its design is an adjustable feature which is infinitely variable for a full* 1.27 cm. This will compensate for uneven ceiling heights and allow adjustment of the sprinkler cover at any time. The feature will also allow the removal of the cover plate to facilitate the changing of soiled or damaged ceiling tiles without the removal of the sprinkler head. This provides significant savings for building maintenance.

The cover plate is designed to fall away during a fire and expose the concealed sprinkler to the rising temperatures of the fire. The deflector drops down to a point below the ceiling line. The heat of the fire then operates the glass bulb sprinkler and water is discharged onto the fire.

2.2.1.2 Technical Description

a) Temperature ratings
   155°F (68°C) Sprinkler, 135°F (57°C) Cover Plate
   200°F (93°C) Sprinkler, 165°F (74°C) Cover Plate
b) Water Working Pressure Rating - 175 psi (12 Bars)
c) Factory tested hydrostatically to 500 psi (34 Bars)
d) Maximum low temperature glass bulb rating is -67°F (-55°C)
e) Frame – brass   Deflector – bronze   Screw – brass
f) Bulb seat – copper   Spring - nickel alloy   Seal – Teflon
g) Bulb - glass with glycerin solution, 5mm

2.2.1.3 Specification of Sprinkler

a) Orifice size: 1/2" (15mm)
b) K Factors: 5.6 (80 metric)
c) Thread Size: 1/2" NPT
d) Finishes: Bright Chrome White Printed

2.2.2 Sprinkler System Design Requirement

2.2.2.1 Design Standard

Under the Uniform Building By-laws 1984, By-laws 226 and 228 refer to the requirement for sprinkler systems. The accepted standards for automatic sprinkler installations are:

a) LPC Rules for Automatic Sprinklers, UK
b) B.S.5306: Part 2- Specification for Sprinkler systems

In the addition to the above, the other standards may be accepted by the Fire and Rescue Department Malaysia but prior approval must be obtained. Some of the standards which have been accepted are:

a) NFPA 13
b) Australian STD A.S. 2118
c) Factory Mutual

Sprinkler systems are designed based on the hazard classification described in B.S. 5306: Part 2 as follows:

a) Light Hazard for non-industrial occupancies with low quantity and combustible contents. For example: apartments, schools and home.

b) Ordinary Hazard for commercial and industrial occupancies handling and storing ordinary combustible materials. For example: Offices, restaurant and hotels.
c) High Hazard for commercial and industrial occupancies having abnormal fire loads covering process hazards, high piled storage hazards and oil and flammable liquid hazards. For example: Furniture and textiles factories.

For this project case, the sprinkler system should be design according to Light Hazard.

2.3 Computational Fluid Dynamics (CFD)

CFD is the systematic application of computing systems and computational solution techniques to mathematical models formulated to describe and simulate fluid dynamic phenomena.

CFD is part of computational mechanics, which in turn is part of simulation techniques. Simulation is used by engineers and physicists to forecast or reconstruct the behavior of an engineering product or physical situation under assumed or measured boundary conditions (geometry, initial states, loads, etc.).

CFD also is a computational technology that enables us to study the dynamics of things that flow. Using CFD, we can build a computational model that represents a system or device that we want to study. Then we apply the fluid flow physics and chemistry to this virtual prototype, and the software will output a prediction of the fluid dynamics and related physical phenomena. Therefore, CFD is a sophisticated computationally-based design and analysis technique. CFD software gives us the power to simulate flows of gases and liquids, heat and mass transfer, moving bodies, multiphase physics, chemical reaction, fluid-structure interaction and acoustics through computer modeling. Using CFD software, we can build a 'virtual prototype' of the system or device that we wish to analyze and then apply real-world physics and chemistry to the model, and the software will provide us with images and data, which predict the performance of that design.

To run a simulation using CFD, the lecture room's boundary condition must be figured out first.

2.3.1 Lecture Room's Boundary Conditions

A general room has heat and flow boundary that is as follows:

2.3.1.1 Air Conditioner

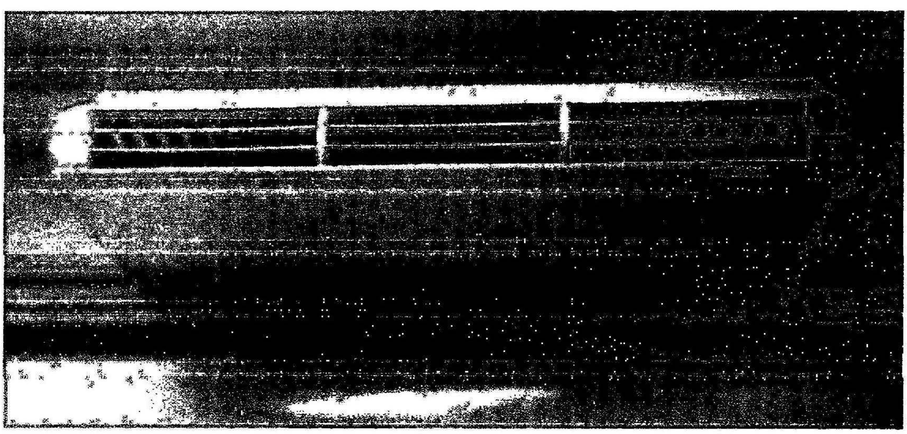


Figure 2.4: Air Conditioner (**YORK** Prestige Ceiling Exposed)

The function of this conditioner is to cool the room. When this air conditioner operate, it produce air flow rate. This air flow rate is one of the boundary condition that needs to figure out for CFD analysis. There are two air conditioners that used in the room. Both of them are from York's model but have different power. Technical descriptions about these air conditioners are below:

Model 1: YMSD1020A (2HP)
Power Source: 220-240 V/50 Hz
Compressor Type: Rotary
Input Power: 1971 W
Casing Material: Galvanized Mild Steel

Model 2: YMST1515125A (2.5HP)
Power Source: 220-240 V/50 Hz
Compressor Type: Rotary
Input Power: 2430 W
Casing Material: Galvanized Mild Steel

2.3.1.2 Lamps

Figure 2.5: Lamp

There are 12 lamps that used in the room. Each one produces 60 Watts.

2.3.1.3 Personal Computer

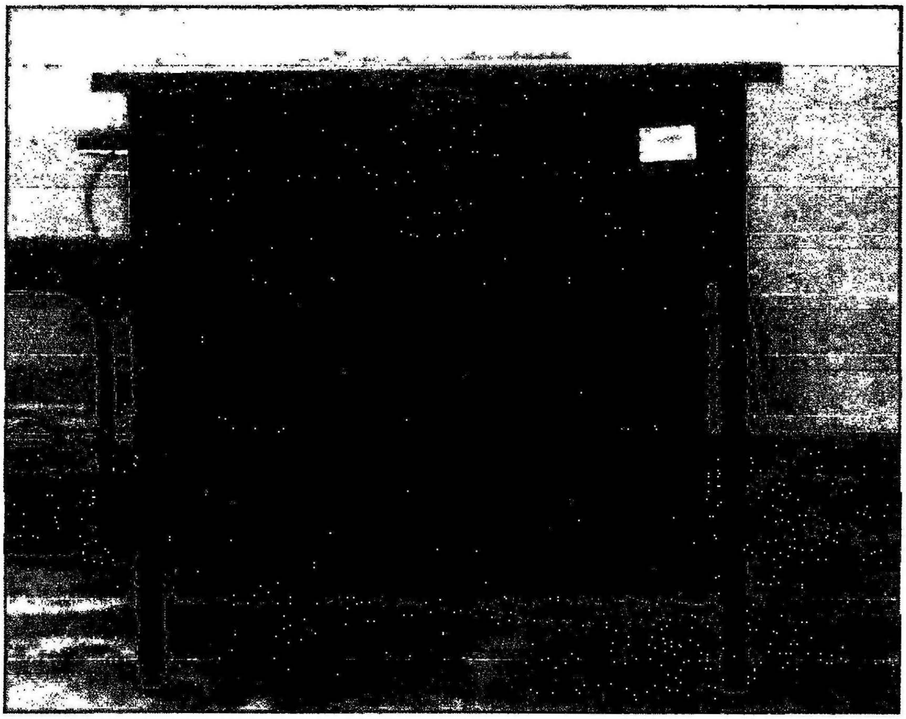


Figure 2.6: Personal Computer (PC)

Desktop or personal computer (PC) that used is from Dell's model. It produces 230 Watts.

2.3.1.4 Wall and Window

These two things are assumed as adiabatic (heat insulated walls) that means there are no heat transfer activities (Yunus A. Cengel, 2002).

2.3.2 CFD Flow Chart

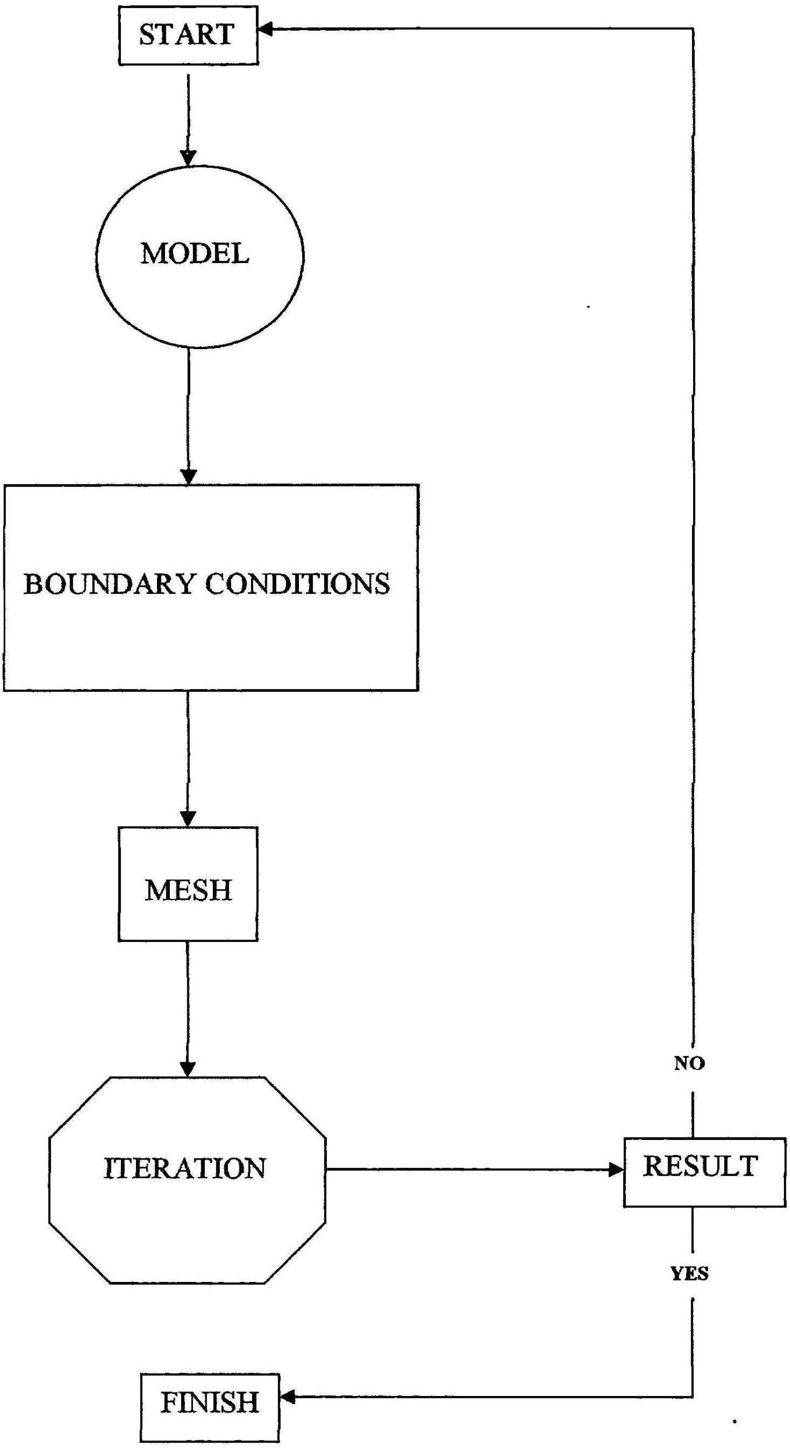


Figure 2.5: CFD Flow Chart

CHAPTER 3

METHODOLOGY

3.1 Problem Solving Method

In order to propose a design of sprinkler system based of CFD analysis of the lecture room there are a few requirements needed. Firstly, to run a CFD analysis we must build a room simulation model using SOLIDWORK and actual measurement of that room must be taken. Then, the boundary condition of that room such as mass flow rate by air conditioner, PC and lamp's power output must be figure out.

3.2 Activities In Each Methodology

3.2.1 Modeling

After taking actual measurement of the lecture room, the 3D room model is designed using SOLIDWORK. Below is the room model:

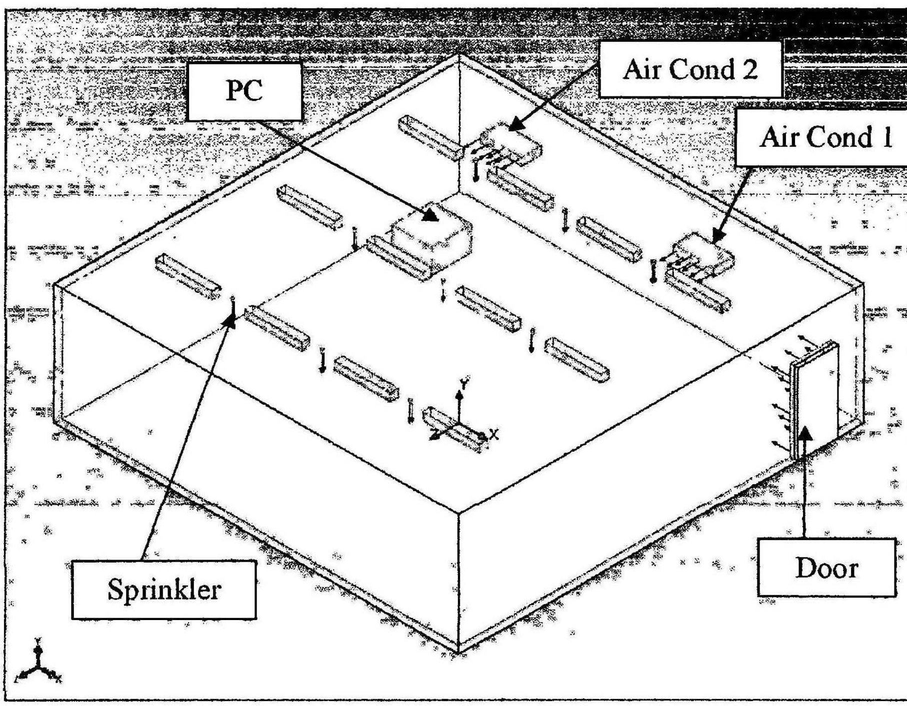


Figure 3.1: Lecture Room (Block Y-BK8) 3D Model

3.2.2 CFD (Computational Fluid Dynamics)

This computational software can build a virtual model that represents a lecture room for study. When the boundary condition (fluid flow physics such as flow rate by air conditioner) is applied to this virtual room prototype, the software will output a prediction of the fluid dynamics and related physical phenomena. The CFD's software that used for this analysis is COSMOSFLOWORKS.

3.2.3 Anemometer

This device is used to measure air conditioner output velocity in order to get its mass flow rate. The mass flow rate then is apply through CFD for analysis of the room.

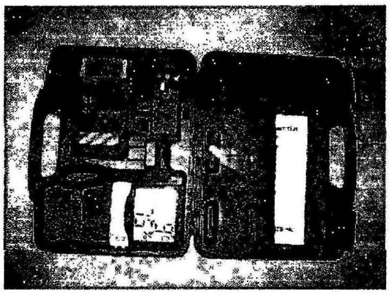


Figure 3.2: Anemometer

3.2.4 Measurement Tape

For taking room actual measurement, this device is used.

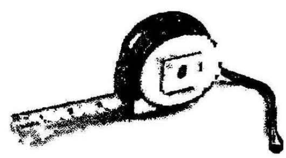


Figure 3.3: Measurement Tape

CHAPTER 4

RESULT AND DISCUSSION

4.1 Introduction

The main objective of this work was to study the effect of sprinkler using CFD for simulation. As a comparison, simulation of the room without sprinkler was also made. Before the simulation can be done, boundary conditions of the room must be found to fulfill the requirement for running the simulation. These are such as, flow rate of the air conditioner and sprinkler.

This simulation will prove the effect of sprinkler in order to suppress fire due to short-circuiting on a PC (personal computer).

4.2 Mass Flow Rate of the Air Conditioner

There are two air conditioners that are used in the lecture room. Both of them were to be from the same brand but have different power outputs. By using anemometer, velocity of the air flow from the air conditioner can be measured. Firstly, both air conditioners are set at fully maximum condition and readings were taken when steady state condition is arrived. Below is the result from this experiment:

Table 4.1: Velocity of the Air Conditioner

No	Air Conditioner 1 (2 HP)	Air Conditioner 2 (2.5 HP)
1	2.0 m/s	3.4 m/s
2	2.1 m/s	3.5 m/s
3	2.1 m/s	3.5 m/s
4	2.1 m/s	3.4 m/s
Average	2.075 m/s	3.45 m/s

To determine the flow rate the equation used is:

$$\dot{m} = \rho A V \tag{4.1}$$

With, ρ=1.1674 kg/m^3 is taken from Table A.4 (Thermo physical Properties of Gases at Atmospheric Pressure) when temperature at 25.7 ^{0}C (298.7 K) and area of the both air conditioner, A= 0.1*1.10 = 0.11 m^2

So the flow rate is given in table 4.2.

Table 4.2: Mass Flow Rate of Air Conditioner

Air Conditioner	Mass Flow Rate, $\dot{m}$ (kg/s)
2 HP	0.2665
2.5 HP	0.443

4.3 Sprinkler's Air Velocity

The simulation has two types which is fluid type and gas type. The sprinkler discharge water about 113.65 liters per minute. But if we use fluid type, the room in the simulation will contain water from sprinkler and air conditioner. So, as a solution, gas type's simulation is chosen and water discharge from sprinkler is converted to air using Reynolds similarity equation.

$$\mathrm{Re}_{water} = \mathrm{Re}_{air}$$

As we know Reynolds equation;

$$Re = \frac{\text{Dynamic pressure}}{\text{Shearing stress}} = \frac{\rho v_s^2/L}{\mu v_s/L^2} = \frac{\rho v_s L}{\mu} = \frac{v_s L}{\nu} = \frac{\text{Inertial forces}}{\text{Viscous forces}}$$

Figure 4.1: Reynolds Equation

Where;

v_s is the mean fluid velocity in ms^{-1}

L is the characteristic length in m

μ is the (absolute) dynamic fluid viscosity in Nsm^{-2} or Pa·s

ν is the kinematics fluid viscosity, defined as $\nu = \mu/\rho$, in m^2s^{-1}

ρ is the density of the fluid in kgm^{-3}

At T= 293.2 K;

From the Table A-4, the results are shown below:

L= length of sprinkler

$Q=VA$

V_{water}= (Q/A) sprinkler

=1.89417/0.0042

= 451 m/s

Table 4.3: Properties of Sprinkler for Water and Air

	WATER	AIR
Density, ρ (kgm^{-3})	0.6238	726.38
Viscosity, v (in m^{2}s^{-1})	2.91 X 10^{-5}	8.872 X 10^{-5}
Velocity, v_s(ms^{-1})	451	1.1808

$$\left(\frac{density * velocity}{viscosity}\right)water = \left(\frac{density * velocity}{viscosity}\right)air$$

V_{air}= {0.6238*451/2.91*10^{-5}}*(8.872*10^{-5})/726.38
= 1.1808 m/s

4.4 Simulation Results

The simulation results shows the final result for both condition tested, where the first condition is where the lecture room model without sprinkler and second condition is where the lecture room model with sprinkler.

4.4.1 Simulation Result without Sprinkler

During simulation there are no sprinkler activated. The purpose of this simulation is to study the heat or temperature distribution in the room at room temperature.

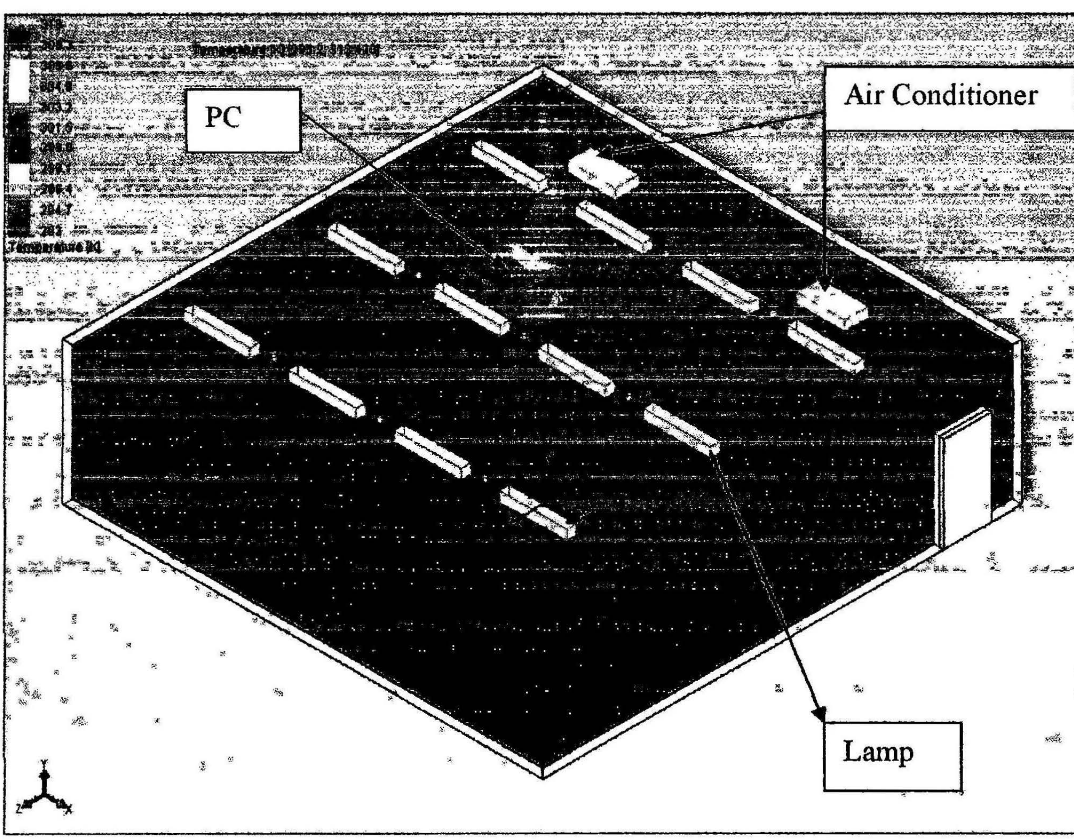


Figure 4.2: Simulation for lecture room without sprinkler

From simulation result in Figure 4.2, it shows that the highest temperature located at the PC which the location that produce higher power output, 230 W. But overall temperature at the room is at room temperature, 293.2 K. There is also a temperature change at the wall close to the PC due to the heat output from PC. The distribution is more at the PC as the temperature increases due to PC operations. Looking at Figure 4.3 the final highest temperature is seen on the PC it self making it a high possibility to ignite in case of some fire in the room.

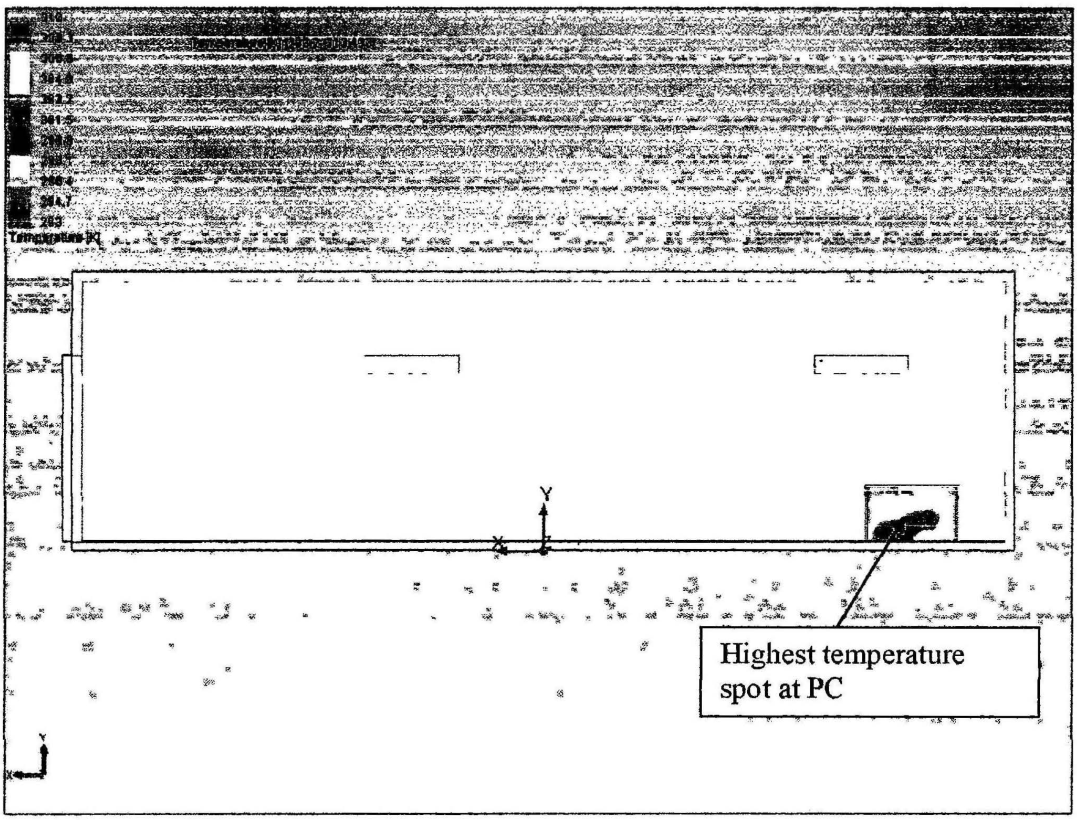


Figure 4.3: Highest temperature spot at Personal Computer (PC)

To identify the highest temperature that spotted at the PC. A line graph plot is sketched as Figure 4.4, to create a Temperature versus Length graph like Figure 4.5.

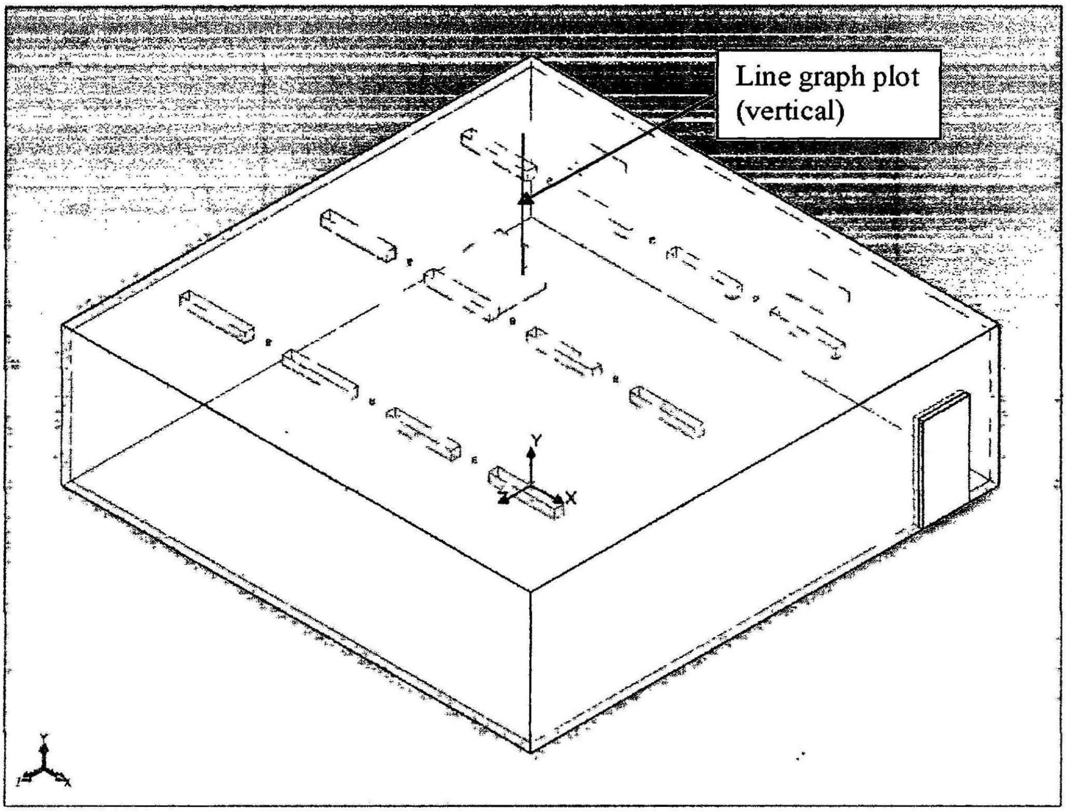


Figure 4.4: Line Graph Plot (vertical) at PC

Figure 4.5: Temperature Distribution-Length Graph

From the graph, the highest temperature detected is around 301 K at PC. The room's temperature is at 293.2 K. But overall result show above it. The lowest temperature is at 294.5 K (close to room's ceiling). This is because of the heat transfer spread due to PC's operation.

The result in Figure 4.5, show the heat transfer from PC to ceiling of the room through a vertical line in Figure 4.4. Then, to study heat transfer from PC to door, a line graph plotted horizontally as Figure 4.7.

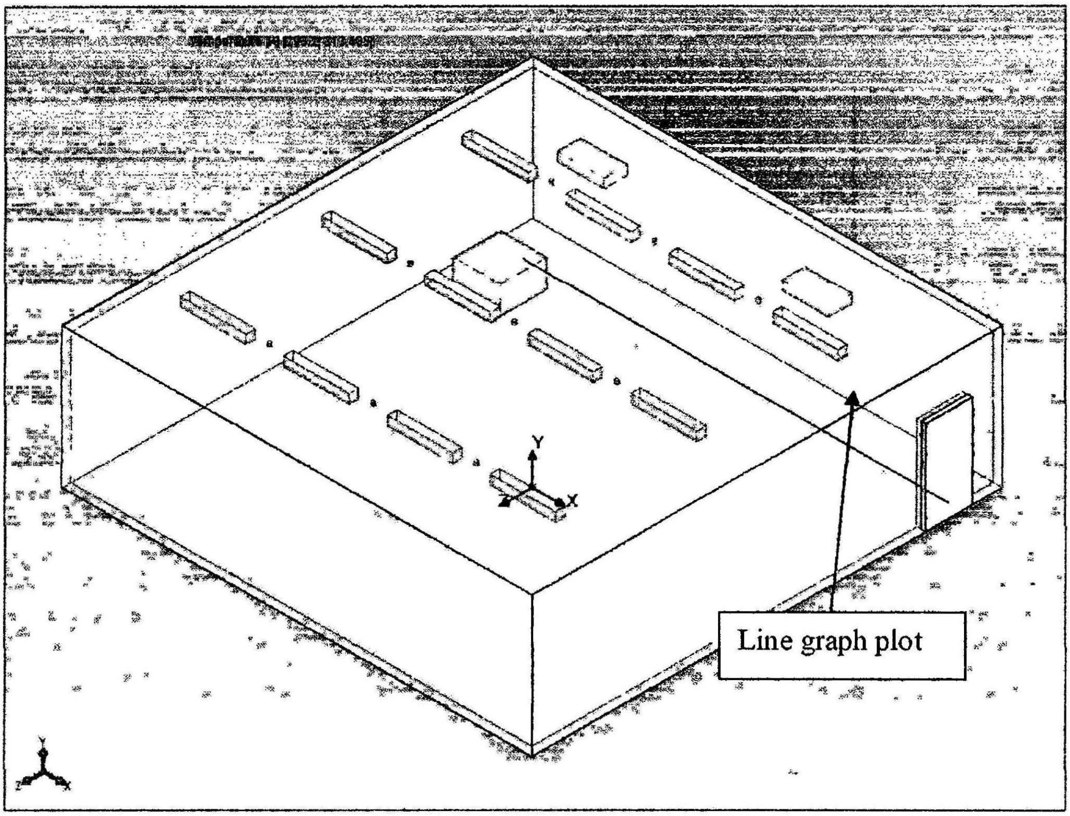


Figure 4.6: Line Graph Plot (horizontal)

From the line in Figure 4.6, a graph likes Figure 4.7 come as a result.

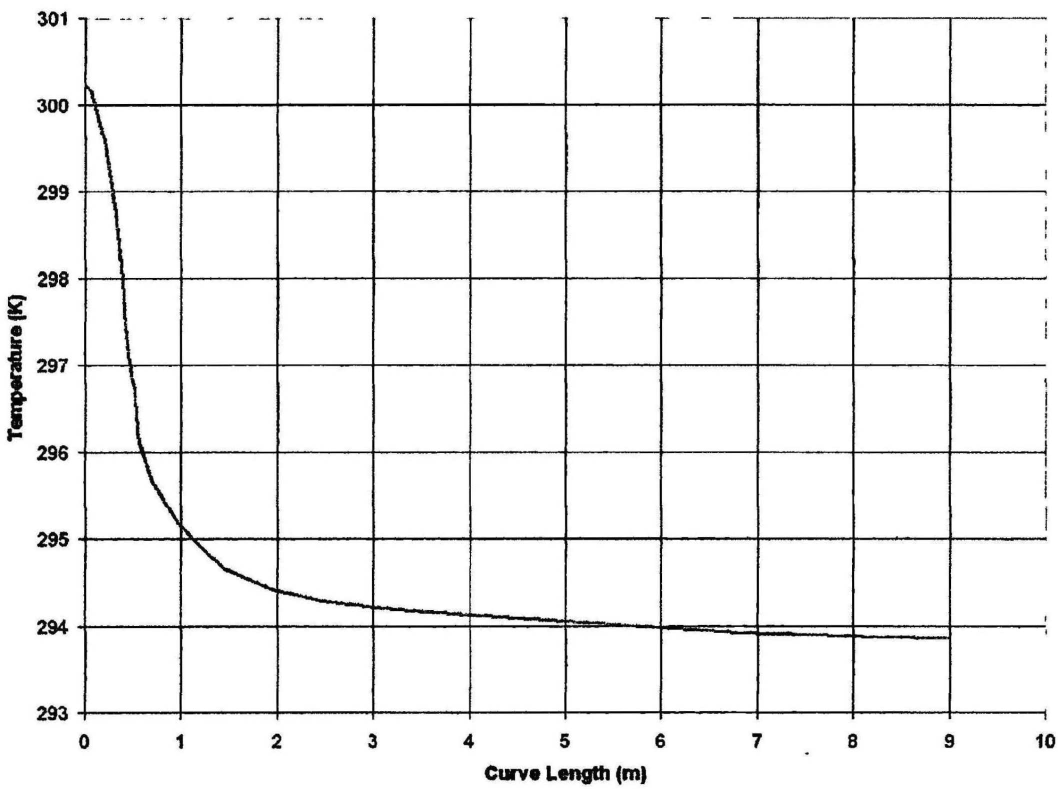


Figure 4.7: Temperature Distribution-Length Graph (horizontal line)

From the result in Figure 4.7, the highest temperature start at PC around 300 K and decreased to 293.9 K (at the door). Overall temperature is above room temperature, 293.2 K but it just a slight different cause by heat transfer spread due to PC's operation.

Actually PC's temperature at 300 K not a critical temperature, but as a precaution that PC should not be operate at long time and must be shut down if no need to used.

4.4.2 Simulation result with sprinkler

For this simulation, assume the fire occurred due to electric shock circuit at PC. Before simulation, as boundary conditions, PC's temperature is set at 341 K which is the temperature that will activate the sprinkler to discharge water. Normally the fire temperature is higher than that but in case of sprinkler; at 341 K it will start operating.

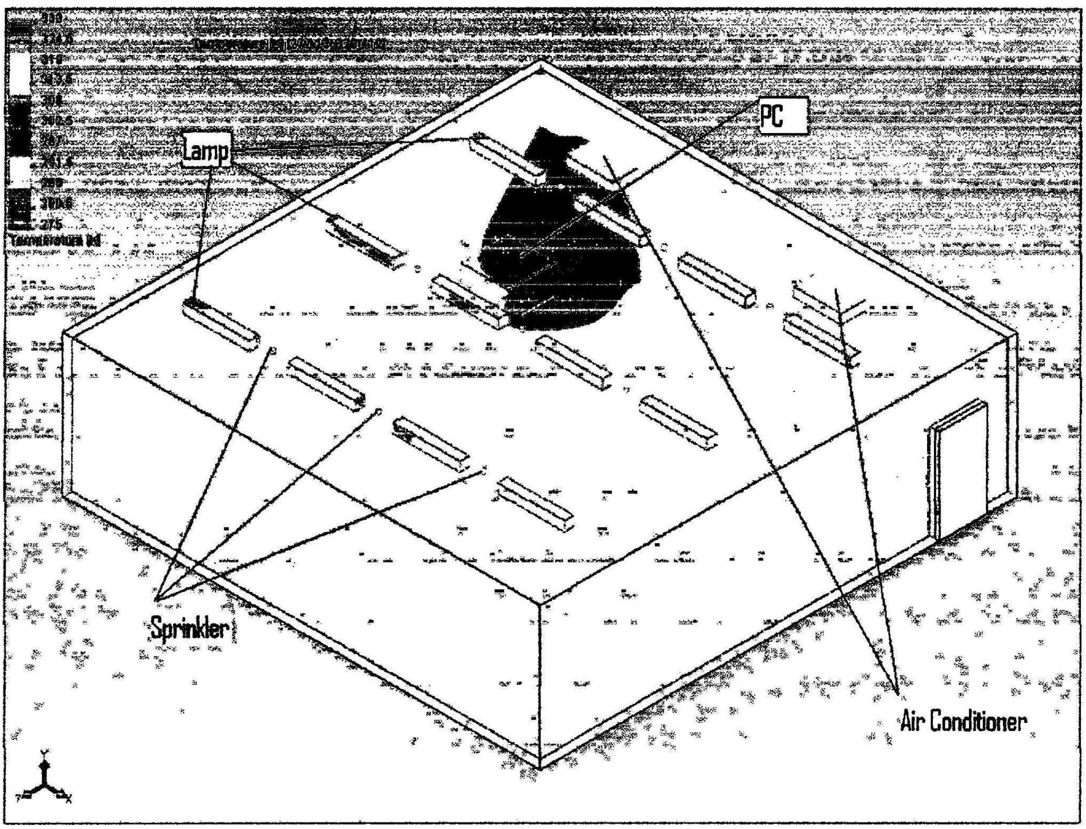


Figure 4.8: Simulation for Lecture Room with Sprinkler

From the simulation result, the PC's temperature is decreased from 341 K to 297 K. The room temperature is 293.2 K. It proves the effect of sprinkler to suppress the fire. But, there are still a bit of higher temperature spot at PC.

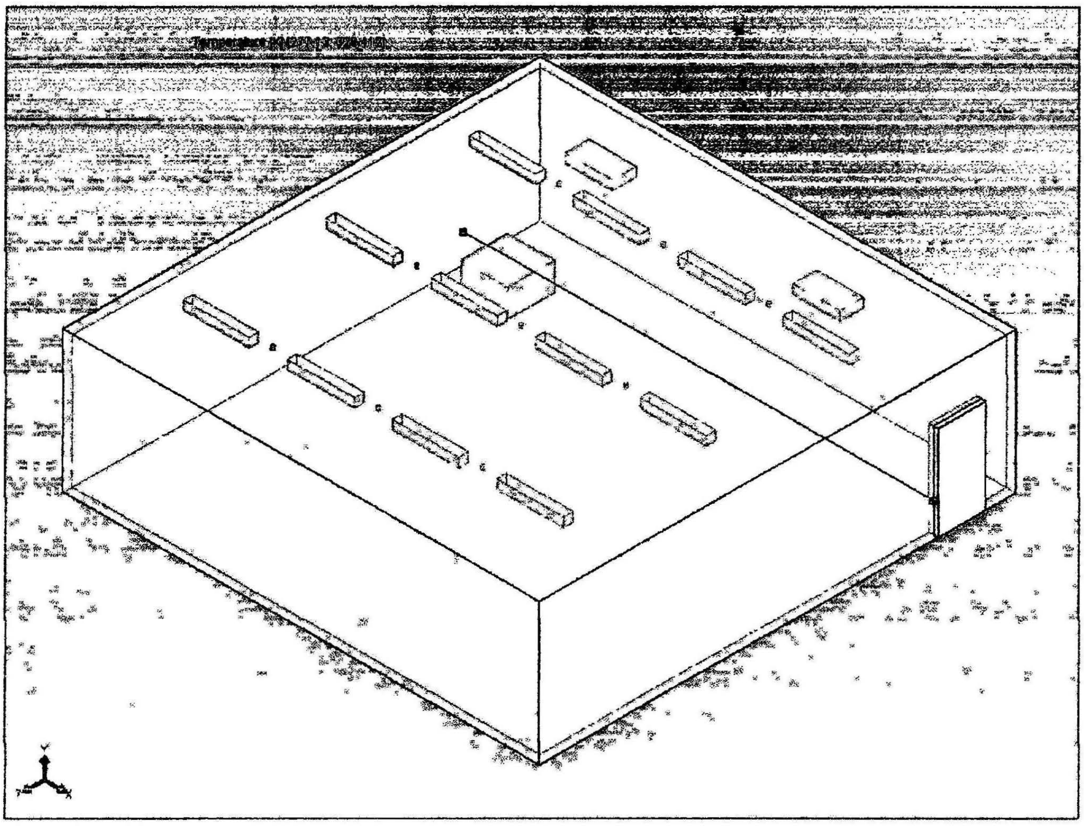


Figure 4.9: Line Graph Plot for Simulation with Sprinkler

To analyze the temperature distribution in the room, one line graph plot across room is created like Figure 4.9. Then, graph Temperature versus Length like Figure 4.10 below is plotted.

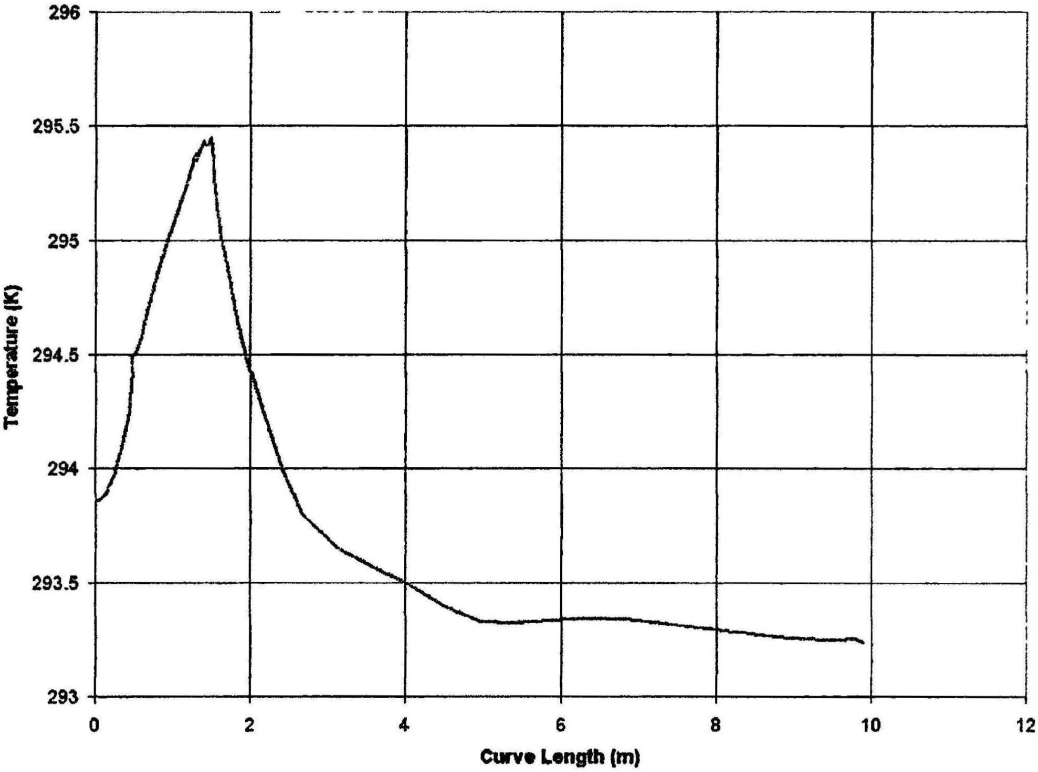


Figure 4.10: Temperature-Length Graph (with Sprinkler)

From the graph, the PC's top surface temperature (green color area) in Figure 4.7 is at 295.4 K. It is higher than the room temperature, which is at 293.2 K but. In this simulation, to create a fire simulation at PC, all PC's surface is set at 341 K which is the temperature that will activate the sprinkler. Overall room temperature is at 293.3 K and it proves the effect of the sprinkler to contain or put out a fire.

Another line graph plot is created to analyze temperature distribution in the room like *Figure 4.11* below.

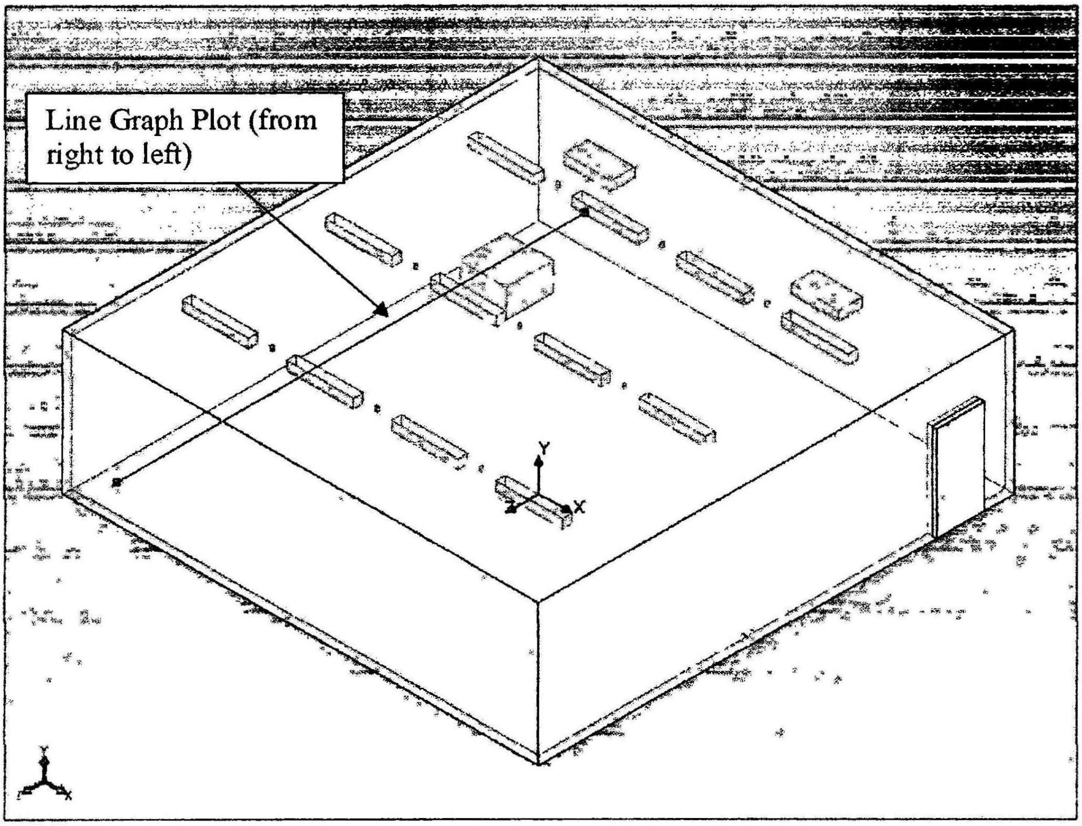


Figure 4.11: Line Graph Plot for Room with Sprinkler

Then, from the line in Figure 4.11, a graph as Figure 4.12 is produced.

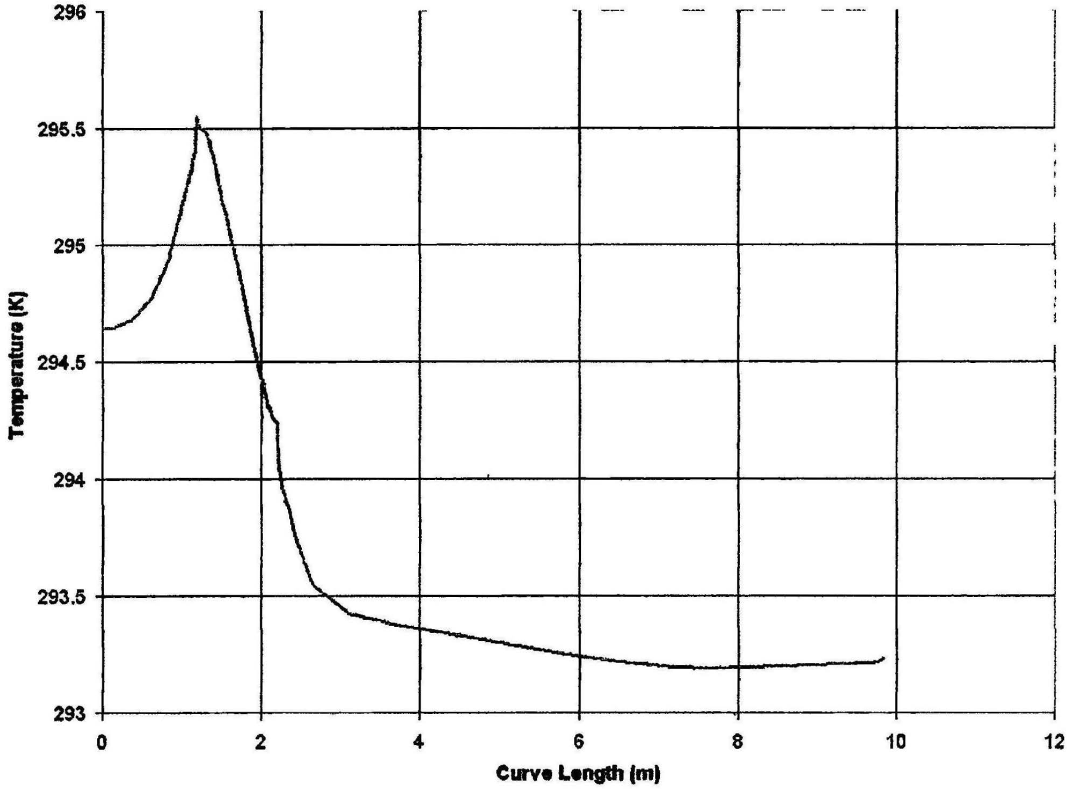


Figure 4.12: Temperature-Length Graph 2 (with sprinkler)

From the graph, the temperature only increased close to the PC. The highest temperature detected is around 295.5 K and only a slight different than room temperature, 293.2 K. Temperature that activates the sprinkler is 341 K which is set at PC to create a fire simulation. Then, as a comparison, result that shows in Figure 4.12, prove the effect of sprinkler to contain a fire. The overall temperature is around 293.3 K.

CHAPTER 5

CONCLUSION AND RECOMMENDATION

5.1 Conclusion

Sprinkler system is the most efficient methods of controlling and suppressing a fire. According to the NFPA, sprinklers typically reduce your chances of dying by one-half to two-thirds in any kind of property where they are used. Despite what many people think, generally only one or two sprinklers open up and flow water during fire-all of them do not go off at once.

Currently, there is no sprinkler system installed at lecture room in UMP (Universiti Malaysia Pahang). Using CFD (Computational Fluid Dynamics), a simulation of fire is created to prove the effect of sprinkler. But before that, one lecture room is chosen for this project (Block Y-BK 8). Using a measurement tape, that room is measured then a 3D model is designed using SOLIDWORK.

The simulation need a room's boundary conditions to run such as power output of PC and lamps, air conditioner flow rate and sprinkler flow rate. The wall of the room is assumed as adiabatic. That mean no heat transfer that flow from inside and outside room.From an experiment using anemometer and some calculation, air conditioner's mass flow rate can be found. Simulation has two type which are gas type and fluid type. If fluid type is chosen, all room will filled with water from sprinkler and air conditioner.

So, as a solution gas type simulation is used. Sprinkler discharged water at certain amount. Reynolds similarity equation is used to calculate sprinkler's air velocity. Then, simulation can be run.

After all the simulation data and analysis is completed, all the objective had archived, all the scope has successfully fulfill. This project become one of the research to prove the efficiency of sprinkler to put out fire.

5.2 Recommendation

As for the future researches, additional element can be carried out to improve the result of the simulation such as;

- Add material to the model. For example, PC is made from aluminum.
- Find out wall's heat transfer rate and not assume it as adiabatic.
- Build a real model of the room including chair and table.
- Use other Computational Fluid Dynamics (CFD) software such as FLUENT to do the same analysis and compare the result.

REFERENCES

Incropera, DeWill, Bergman, Lavine(2000), *Introduction To Heat Transfer*. Fifth Edition, John Wiley & Sons (Asia) Pte. Ltd

Yunus A. Cengel, Michael A. Boles (2002), *Thermodynamics: An Engineering Approach, Fourth Edition in SI Units*, McGraw-Hill Education(Asia)

Prof Datuk Dr Soh Chai Hock (1998),*Guide To Fire Protection In Malaysia*,Fire and Rescue Department Malaysia, Pertubuhan Arkitek Malaysia (PAM), Institution of Engineer Malaysia (IEM) and Malaysia Fire Protection Association (MFPA)

Writer-Tech.com (2001), *Living With Fire (A Program For Campus And Student Fire Safety)*

Yunus A Cengel,John M. Cimbala, (2006), *Fluid Mechanics Fundamentals and Application*,, Mc Graw Hill,Companies□

APPENDIX A

FULL REPORT (Room without Sprinkler Analysis)

System Info

Product	COSMOSFloWorks 2005/ PE SP0.0. Build: 157
Computer Name	UMP-76972E8C0C8
User Name	W@N
Processors	Intel
Memory	510 MB / 2047 MB
Operating System	Windows XP Service Pack 2
CAD Version	SolidWorks 2005 SP0
CPU Speed	586 (2414 MHz)

General Info

Model	C:\Documents and Settings\W@N\Desktop\PSM 2 Draft\Result\without sprinkler (5.03.2008)\BK8.SLDPRT
Project Name	Default (2)
Project Path	C:\Documents and Settings\W@N\Desktop\PSM 2 Draft\Result\without sprinkler (5.03.2008)\1\1.fwp
Units system	
Analysis type	Internal
Exclude cavities without flow conditions	On
Coordinate system	Global Coordinate System
Reference Axis	X

INPUT DATA

Initial Mesh Settings

Automatic Initial Mesh: On
Result resolution level: 3
Advanced narrow channel refinement: Off
Refinement in solid region: Off

Geometry resolution

Evaluation of minimum gap size: Automatic
Evaluation of minimum wall thickness: Automatic

Computational Domain

Size

X min	-5 m
X max	5.1 m
Y min	0 m
Y max	3 m
Z min	-5 m
Z max	5 m

Boundary Conditions

2D plane flow	None
At X min	Default
At X max	Default
At Y min	Default
At Y max	Default
At Z min	Default
At Z max	Default

Physical Features

Heat transfer in solids: Off
Time dependent: Off
Gravitational effects: Off
Flow type: Laminar and Turbulent
High Mach number flow: Off
Default roughness: 0 micrometer
Default wall conditions: Adiabatic wall

Initial Conditions

Thermodynamic parameters	Static Pressure: 101325 Pa Temperature: 293.2 K
Velocity parameters	Velocity vector X component of velocity: 0 m/s Y component of velocity: 0 m/s Z component of velocity: 0 m/s
Turbulence parameters	Turbulence intensity and length Intensity: 2 % Length: 0.000541670243 m

Material settings

Fluid type: Gas

Fluids

Air

Boundary Conditions

Static Pressure1

Type	Static Pressure
Faces	Face <84 >
Coordinate system	Face based coordinate system
Reference Axis	X
Thermodynamic parameters	Static Pressure: 101325 Pa Temperature: 293.2 K
Turbulence parameters	Turbulence intensity and length Intensity: 2 % Length: 0.000541670243 m
Boundary layer parameters	Boundary layer type: Turbulent

Inlet Mass Flow1

Type	Inlet Mass Flow
Faces	Face <53 >
Coordinate system	Face based coordinate system
Reference Axis	X
Flow parameters	Flow vectors direction: Normal to face Mass flow rate normal to face: 0.2665 kg/s Fully developed tube flow: No Inlet profile: 0
Thermodynamic parameters	Temperature: 293.2 K
Turbulence parameters	Turbulence intensity and length Intensity: 2 % Length: 0.000541670243 m
Boundary layer parameters	Boundary layer type: Turbulent

Inlet Mass Flow2

Type	Inlet Mass Flow
Faces	Face <62 >
Coordinate system	Face based coordinate system
Reference Axis	X
Flow parameters	Flow vectors direction: Normal to face Mass flow rate normal to face: 0.443 kg/s Fully developed tube flow: No Inlet profile: 0
Thermodynamic parameters	Temperature: 293.2 K
Turbulence parameters	Turbulence intensity and length Intensity: 2 % Length: 0.000541670243 m
Boundary layer parameters	Boundary layer type: Turbulent

Heat surface sources

SS Heat Transfer Rate 2	
Type	Heat Transfer Rate
Faces	Face <18LPattern7>
	Face <15LPattern7>
	Face <4LPattern7>
	Face <14LPattern7>
	Face <5LPattern7>
	Face <17LPattern7>
	Face <12LPattern7>
	Face <8LPattern7>
	Face <16LPattern7>
	Face <13LPattern7>
	Face <3LPattern7>
	Face <2LPattern7>
	Face <6LPattern7>
	Face <9LPattern7>
	Face <19LPattern7>
	Face <10LPattern7>
	Face <11LPattern7>
	Face <7LPattern7>
	Face <1LPattern7>
Coordinate system	Global Coordinate System
Reference Axis	X
Heat Transfer Rate	60 W
SS Heat Transfer Rate 3	
Type	Heat Transfer Rate
Faces	Face <14Extrude8>
	Face <12LPattern6>
	Face <20LPattern6>
	Face <19Extrude8>
	Face <5LPattern6>
	Face <3Extrude8>
	Face <11Extrude8>
	Face <1LPattern5>
	Face <6Extrude8>
	Face <2LPattern5>
	Face <18LPattern5>
	Face <16LPattern5>
	Face <17LPattern5>
	Face <8LPattern5>
	Face <7LPattern5>
	Face <15LPattern5>
	Face <4LPattern6>
	Face <13LPattern6>

	Face <10LPattern5>
	Face <9LPattern5>
Coordinate system	Global Coordinate System
Reference Axis	X
Heat Transfer Rate	60 W
SS Heat Transfer Rate 1	
Type	Heat Transfer Rate
Faces	Face <5Extrude4>
	Face <4Extrude4>
	Face <1Extrude4>
	Face <2Extrude4>
	Face <3Extrude4>
Coordinate system	Global Coordinate System
Reference Axis	X
Heat Transfer Rate	230 W
SS Heat Transfer Rate 4	
Type	Heat Transfer Rate
Faces	Face <19LPattern8>
	Face <18LPattern8>
	Face <15LPattern8>
	Face <5LPattern8>
	Face <20LPattern8>
	Face <14LPattern8>
	Face <4LPattern8>
	Face <6LPattern8>
	Face <13LPattern8>
	Face <8LPattern8>
	Face <17LPattern8>
	Face <16LPattern8>
	Face <12LPattern8>
	Face <3LPattern8>
	Face <2LPattern8>
	Face <9LPattern8>
	Face <11LPattern8>
	Face <7LPattern8>
	Face <1LPattern8>
	Face <10LPattern8>
Coordinate system	Global Coordinate System
Reference Axis	X
Heat Transfer Rate	60 W

Calculation control options

Finish Conditions

Finish Conditions	If one is satisfied
Maximum iterations	202
Maximum travels	4
Goals convergence	Analysis interval: 0.5

Solver Refinement

Refinement: Disabled

Results saving

Save before refinement	On

Advanced control options

Flow Freezing	
Flow Freezing Strategy	Disabled

RESULTS

General Info

Iterations: 149

Calculation Mesh

Basic Mesh Dimensions

Number of cells in X	20
Number of cells in Y	6
Number of cells in Z	20

Number of Cells

Total Cells	11983
Fluid Cells	5441
Solid Cells	341
Partial Cells	6201
Irregular Cells	0

Maximum refinement level: 3

Min/Max Table

Name	Minimum	Maximum
Pressure [Pa]	101324	101326
Temperature [K]	293.2	313.433
Density [kg/m^3]	1.12601	1.20371

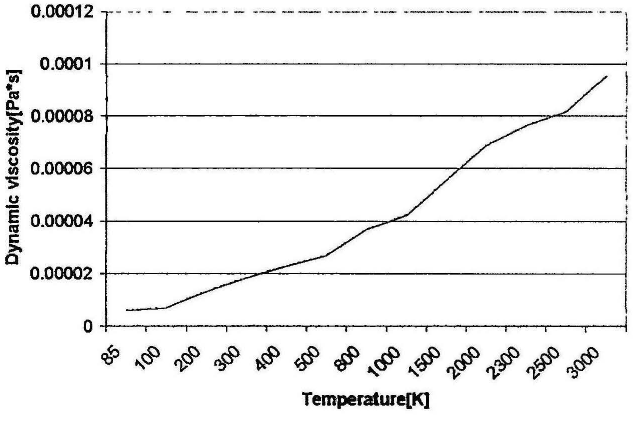
Velocity [m/s]	0	1.88769
X-velocity [m/s]	-0.426749	0.391697
Y-velocity [m/s]	-0.288248	0.297079
Z-velocity [m/s]	-0.32251	1.88713
Heat Transfer Coefficient [w/m^2/K]	0	51.6869
Shear Stress [Pa]	1.08569e-011	0.00510355
Heat Flux [W/m^2]	0	73.6454
Air Mass Fraction [ ]	1	1
Air Volume Fraction [ ]	1	1
Fluid Temperature [K]	293.2	313.433

Engineering Database

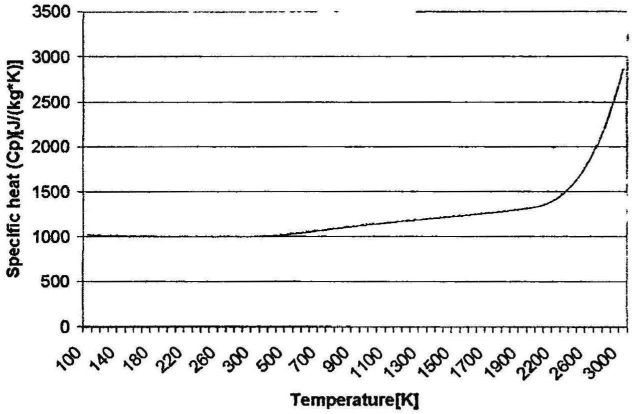
Gases

Air

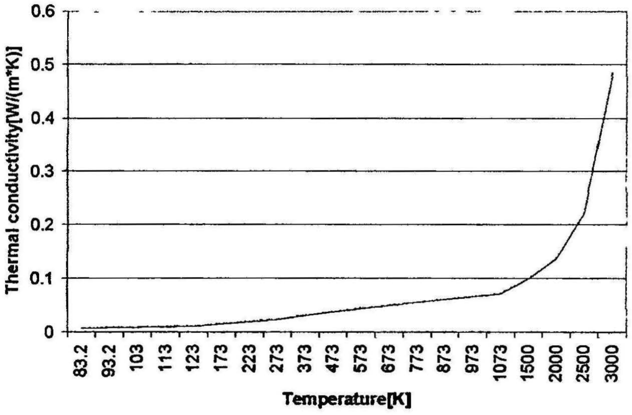
Path: Gas FW Defined
Specific heat ratio (Cp/Cv): 1.399
Molecular mass: 0.02896 kg/mol
Dynamic viscosity



Specific heat (Cp)



Thermal conductivity



APPENDIX B

FULL REPORT (Room with Sprinkler Analysis)

System Info

Product	COSMOSFloWorks 2005/ PE SP0.0. Build: 157
Computer Name	UMP-76972E8C0C8
User Name	W@N
Processors	Intel
Memory	510 MB / 2047 MB
Operating System	Windows XP Service Pack 2
CAD Version	SolidWorks 2005 SP0
CPU Speed	586 (2414 MHz)

General Info

Model	C:\Documents and Settings\W@N\Desktop\PSM 2 Draft\Result\with sprinkler (8.04.2008)\BK8.SLDPRT
Project Name	Default (2)
Project Path	C:\Documents and Settings\W@N\Desktop\PSM 2 Draft\Result\with sprinkler (8.04.2008)\1\1.fwp
Units system	SI (m-kg-s)
Analysis type	Internal
Exclude cavities without flow conditions	On
Coordinate system	Global Coordinate System
Reference Axis	X

INPUT DATA

Initial Mesh Settings

Automatic Initial Mesh: On
Result resolution level: 3
Advanced narrow channel refinement: Off
Refinement in solid region: Off

Geometry resolution

Evaluation of minimum gap size: Automatic
Evaluation of minimum wall thickness: Automatic

Computational Domain

Size

X min	-5 m
X max	5.1 m
Y min	0 m
Y max	3 m
Z min	-5 m
Z max	5 m

Boundary Conditions

2D plane flow	None
At X min	Default
At X max	Default
At Y min	Default
At Y max	Default
At Z min	Default
At Z max	Default

Physical Features

Heat transfer in solids: Off
Time dependent: Off
Gravitational effects: Off
Flow type: Laminar and Turbulent
High Mach number flow: Off
Default roughness: 0 micrometer
Default wall conditions: Adiabatic wall

Initial Conditions

Thermodynamic parameters	Static Pressure: 101325 Pa Temperature: 293.2 K
Velocity parameters	Velocity vector X component of velocity: 0 m/s Y component of velocity: 0 m/s Z component of velocity: 0 m/s
Turbulence parameters	Turbulence intensity and length Intensity: 2 % Length: 0.000541670243 m

Material settings

Fluid type: Gas

Fluids

Air

Boundary Conditions

Static Pressure1	
Type	Static Pressure
Faces	Face <61 >
Coordinate system	Face based coordinate system
Reference Axis	X
Thermodynamic parameters	Static Pressure: 101325 Pa Temperature: 293.2 K
Turbulence parameters	Turbulence intensity and length Intensity: 2 % Length: 0.000541670243 m
Boundary layer parameters	Boundary layer type: Turbulent
Inlet Velocity1	
Type	Inlet Velocity
Faces	Face <62 > Face <31 > Face <19 > Face <41 > Face <57 > Face <30 > Face <66 > Face <1 > Face <20 >
Coordinate system	Global Coordinate System
Reference Axis	X
Flow parameters	Flow vectors direction: Normal to face Fully developed tube flow: No Velocity normal to face: 1.1808 m/s Inlet profile: 0
Thermodynamic parameters	Temperature: 277.13 K
Turbulence parameters	Turbulence intensity and length Intensity: 2 % Length: 0.000541670243 m
Boundary layer parameters	Boundary layer type: Turbulent
Inlet Mass Flow1	
Type	Inlet Mass Flow
Faces	Face <44 >
Coordinate system	Face based coordinate system
Reference Axis	X
Flow parameters	Flow vectors direction: Normal to face Mass flow rate normal to face: 0.2665 kg/s

	Fully developed tube flow: No
	Inlet profile: 0
Thermodynamic parameters	Temperature: 293.2 K
Turbulence parameters	Turbulence intensity and length
	Intensity: 2 %
	Length: 0.000541670243 m
Boundary layer parameters	Boundary layer type: Turbulent
Inlet Mass Flow2	
Type	Inlet Mass Flow
Faces	Face <36 >
Coordinate system	Face based coordinate system
Reference Axis	X
Flow parameters	Flow vectors direction: Normal to face
	Mass flow rate normal to face: 0.443 kg/s
	Fully developed tube flow: No
	Inlet profile: 0
Thermodynamic parameters	Temperature: 293.2 K
Turbulence parameters	Turbulence intensity and length
	Intensity: 2 %
	Length: 0.000541670243 m
Boundary layer parameters	Boundary layer type: Turbulent

Heat surface sources

SS Heat Transfer Rate 2	
Type	Heat Transfer Rate
Faces	Face <18LPattern7>
	Face <15LPattern7>
	Face <4LPattern7>
	Face <14LPattern7>
	Face <5LPattern7>
	Face <17LPattern7>
	Face <12LPattern7>
	Face <8LPattern7>
	Face <16LPattern7>
	Face <13LPattern7>
	Face <3LPattern7>
	Face <2LPattern7>
	Face <6LPattern7>
	Face <9LPattern7>
	Face <19LPattern7>
	Face <10LPattern7>
	Face <11LPattern7>
	Face <7LPattern7>
	Face <1LPattern7>
Coordinate system	Global Coordinate System

Reference Axis	X
Heat Transfer Rate	60 W
SS Heat Transfer Rate 3	
Type	Heat Transfer Rate
Faces	Face <14Extrude8>
	Face <12LPattern6>
	Face <20LPattern6>
	Face <19Extrude8>
	Face <5LPattern6>
	Face <3Extrude8>
	Face <11Extrude8>
	Face <1LPattern5>
	Face <6Extrude8>
	Face <2LPattern5>
	Face <18LPattern5>
	Face <16LPattern5>
	Face <17LPattern5>
	Face <8LPattern5>
	Face <7LPattern5>
	Face <15LPattern5>
	Face <4LPattern6>
	Face <13LPattern6>
	Face <10LPattern5>
	Face <9LPattern5>
Coordinate system	Global Coordinate System
Reference Axis	X
Heat Transfer Rate	60 W
SS Heat Transfer Rate 1	
Type	Heat Transfer Rate
Faces	Face <5Extrude4>
	Face <4Extrude4>
	Face <1Extrude4>
	Face <2Extrude4>
	Face <3Extrude4>
Coordinate system	Global Coordinate System
Reference Axis	X
Heat Transfer Rate	230 W
SS Heat Transfer Rate 4	
Type	Heat Transfer Rate
Faces	Face <19LPattern8>
	Face <18LPattern8>
	Face <15LPattern8>
	Face <5LPattern8>
	Face <20LPattern8>

	Face <14LPattern8>
	Face <4LPattern8>
	Face <6LPattern8>
	Face <13LPattern8>
	Face <8LPattern8>
	Face <17LPattern8>
	Face <16LPattern8>
	Face <12LPattern8>
	Face <3LPattern8>
	Face <2LPattern8>
	Face <9LPattern8>
	Face <11LPattern8>
	Face <7LPattern8>
	Face <1LPattern8>
	Face <10LPattern8>
Coordinate system	Global Coordinate System
Reference Axis	X
Heat Transfer Rate	60 W

Initial Conditions

Initial Condition1	
Components	Face <5 Extrude4>
	Face <3 Extrude4>
	Face <1 Extrude4>
	Face <2 Extrude4>
	Face <4 Extrude4>
Coordinate system	Global Coordinate System
Reference Axis	X
Flow parameters	Velocity vector
	X component of velocity: 0 m/s
	Y component of velocity: 0 m/s
	Z component of velocity: 0 m/s
Thermodynamic parameters	Static Pressure: 101325 Pa
	Temperature: 341 K
Turbulence parameters	Turbulence intensity and length
	Intensity: 2 %
	Length: 0.000541670243 m

Calculation control options

Finish Conditions

Finish Conditions	If one is satisfied
Maximum travels	4
Goals convergence	Analysis interval: 0.5

Solver Refinement

Refinement: Disabled

Results saving

Save before refinement	On

Advanced control options

Flow Freezing	
Flow Freezing Strategy	Disabled

RESULTS

General Info

Iterations: 200

Calculation Mesh

Basic Mesh Dimensions

Number of cells in X	22
Number of cells in Y	6
Number of cells in Z	22

Number of Cells

Total Cells	48306
Fluid Cells	17894
Solid Cells	10396
Partial Cells	20016
Irregular Cells	0

Maximum refinement level: 5

Min/Max Table

Name	Minimum	Maximum
Pressure [Pa]	101325	101326
Temperature [K]	277.13	328.412
Density [kg/m^3]	1.07465	1.27351
Velocity [m/s]	0	1.85415
X-velocity [m/s]	-0.327306	0.441034
Y-velocity [m/s]	-1.1808	0.375365
Z-velocity [m/s]	-0.33646	1.85406
Heat Transfer Coefficient [w/m^2/K]	0	263680
Shear Stress [Pa]	3.29589e-012	0.041146
Heat Flux [W/m^2]	0	71.9325

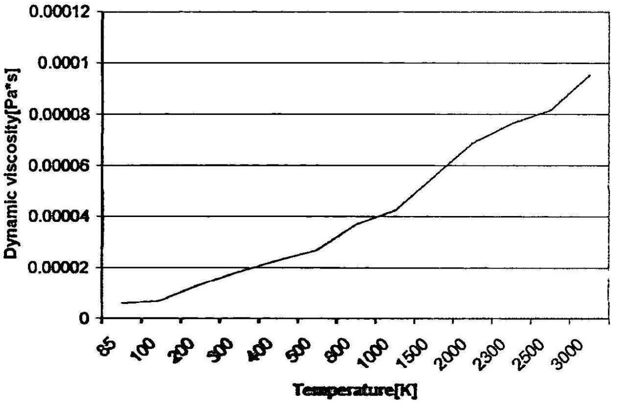
Air Mass Fraction [ ]	1	1
Air Volume Fraction [ ]	1	1
Fluid Temperature [K]	277.13	328.412

Engineering Database

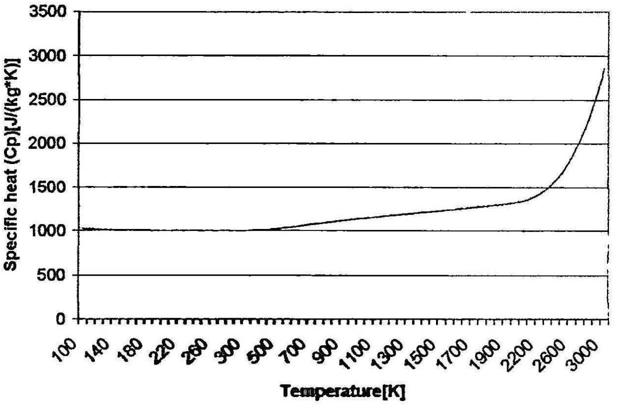
Gases

Air

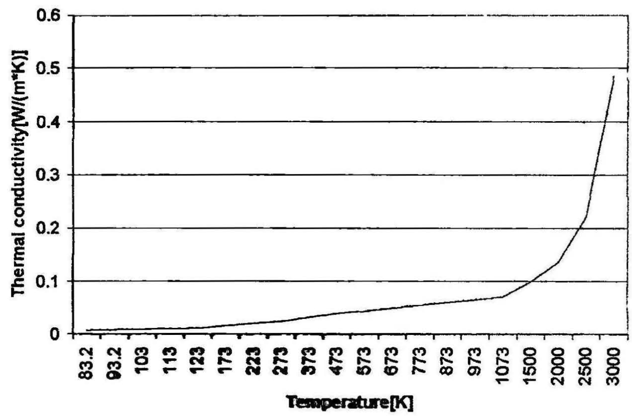
Path: Gas FW Defined
Specific heat ratio (Cp/Cv): 1.399
Molecular mass: 0.02896 kg/mol
Dynamic viscosity



Specific heat (Cp)



Thermal conductivity



APPENDIX C

Lecture Room's Diameter (Blok Y-BK 8)