

**FIRE DYNAMICS SIMULATION (FDS) STUDY OF FIRE IN STRUCTURES
WITH CURVED GEOMETRY**

CHIEW HUI ZHONG

**A project report submitted in partial fulfilment of the
requirements for the award of Bachelor of Engineering
(Hons.) Mechanical Engineering**

**Faculty of Engineering and Science
Universiti Tunku Abdul Rahman**

April 2013

DECLARATION

I hereby declare that this project report is based on my original work except for citations and quotations which have been duly acknowledged. I also declare that it has not been previously and concurrently submitted for any other degree or award at UTAR or other institutions.

Signature : _____

Name : CHIEW HUI ZHONG

ID No. : 09UEB08547

Date : _____

APPROVAL FOR SUBMISSION

I certify that this project report entitled “**FIRE DYNAMICS SIMULATION (FDS) STUDY OF FIRE IN STRUCTURES WITH CURVED GEOMETRY**” was prepared by **CHIEW HUI ZHONG** has met the required standard for submission in partial fulfilment of the requirements for the award of Bachelor of ENGINEERING (Hons.) MECHANICAL ENGINEERING at Universiti Tunku Abdul Rahman.

Approved by,

Signature : _____

Supervisor: Dr. THAN CHEOK FAH

Date : _____

The copyright of this report belongs to the author under the terms of the copyright Act 1987 as qualified by Intellectual Property Policy of Universiti Tunku Abdul Rahman. Due acknowledgement shall always be made of the use of any material contained in, or derived from, this report.

© 2013, Chiew Hui Zhong. All right reserved.

ACKNOWLEDGEMENTS

I would like to thank everyone who had contributed to the successful completion of this project. I would like to express my gratitude to my research supervisor, Dr. THAN CHEOK FAH for his invaluable advice, guidance and his enormous patience throughout the development of the research.

In addition, I would also like to express my gratitude to my loving parent and friends who had helped and given me encouragement in my research.

FIRE DYNAMICS SIMULATION (FDS) STUDY OF FIRE IN STRUCTURES WITH CURVED GEOMETRY

ABSTRACT

The Fire Dynamics Simulator (FDS) is a software that describes a model of fire-driven fluid flow by the application of computational fluid dynamics (CFD). This software is developed by the National Institute of Standard and Technology (NIST) which solves the Navier Stokes equation numerically for low-speed, thermal-driven flow. The FDS produces output which emphasizes on the smoke propagation and fire behaviour. Fire models are treated as turbulent model and to be solved by Large Eddy Simulation (LES). PyroSim with the integration of FDS is used for the fire simulation. The PyroSim uses rectilinear cells in meshing the model before simulation, which results in poor representation on the curved geometry. The refinement of mesh can be done to improve the representation of curved geometry but it may lead to high memory consumption and computational time. Hence, this paper presents to balance the combination of coarse and fine mesh with the curved geometry representation. A few methods of meshing to improve the meshing on curvature surface have been suggested and the simulations are run to determine the suitability of each meshing method. Besides, the simulation is also done using ANSYS FLUENT and the simulation results are compared between FDS and FLUENT. With the integration of Revit Architecture, the structural model can be exported from Revit Architecture to PyroSim for fire simulation. The output of the FDS is then attempted to be exported back into Revit Architecture, which allow user to enter Walkthrough feature to observe the real-life fire scenario.

TABLE OF CONTENTS

DECLARATION	ii
APPROVAL FOR SUBMISSION	iii
ACKNOWLEDGEMENTS	v
ABSTRACT	vi
TABLE OF CONTENTS	vii
LIST OF TABLES	x
LIST OF FIGURES	xi
LIST OF SYMBOLS / ABBREVIATIONS	xv
LIST OF APPENDICES	xvii

CHAPTER

1	INTRODUCTION	1
	1.1 Background	1
	1.2 Theory of CFD	4
	1.3 Aims and Objectives	7
2	LITERATURE REVIEW	9
	2.1 Fire Model Simulation Using FLUENT	9
	2.2 Fire Model Simulation Using FDS	14
	2.2.1 FDS Simulation with Different Meshing Method and Mesh Resolution	14
	2.2.2 FDS Simulation Linkage with other Software	18
	2.2.3 FDS Simulation with Other Fire Safety Devices	21
	2.2.4 FDS Simulation for Fire Reconstruction	22

2.3	Comparison of Simulation Result between FLUENT and FDS	23
2.4	Post-Processing of FDS Simulation Results	26
3	METHODOLOGY	28
3.1	Structural Modelling Using Revit Architecture	28
3.2	Fire Model Simulation in FLUENT	29
3.2.1	Heat Flux Model Using User Defined Function Input	33
3.2.2	Non-premixed Combustion Model Using User Defined Function Input	33
3.3	Fire Model LES Simulation in PyroSim	34
3.3.1	Meshing Methods in FDS	35
3.3.2	Numerical Instability in FDS Simulations	39
3.3.3	Appropriate Mesh Resolution	40
3.3.4	FDS Simulation with Various Curved Geometry	42
3.3.5	FDS Simulation with Various Type of Fire	43
3.4	Fire Model DNS Simulation in PyroSim	44
3.5	Full Scale Model Simulation by FDS and Walkthrough in SmokeView	44
4	RESULTS AND DISCUSSIONS	47
4.1	Fire Model Simulation Using FLUENT	47
4.1.1	Heat Flux Model	47
4.1.2	Boussinesq Heat Flux Model with UDF	48
4.1.3	Non-premixed Combustion Model	50
4.1.4	Non-premixed Combustion Model with UDF	51
4.2	Fire Model LES Simulation Using PyroSim	53
4.2.1	Appropriate Mesh Resolution	54
4.2.2	Meshing Method in FDS	57
4.2.3	Comparison of Simulation Results with FLUENT and FDS	61
4.2.4	FDS Simulation with Various Curved Geometries	68

4.2.5	FDS Simulation with Various Type of Fire	72
4.3	Fire Model DNS Simulation Using PyroSim	74
4.4	Full Scale Model Simulation by FDS and Walkthrough in SmokeView	78
5	CONCLUSION AND RECOMMENDATIONS	83
5.1	Research Summary	83
5.2	Recommendations for Future Research	86
	REFERENCES	88
	APPENDICES	92

LIST OF TABLES

TABLE	TITLE	PAGE
4.1	Comparison of Different Characteristic Fire Diameter Ratio	54
4.2	Comparison of Meshing Methods in FDS	58
4.3	Decision Matrix of Meshing Method Selection	61
4.4	Comparison of Curved Geometry between 1 Mesh and Embedded Mesh	69
4.5	Comparison of Actual CAD Model and FDS Block	78

LIST OF FIGURES

FIGURE	TITLE	PAGE
2.1	Division of Two Way Vent of Model (Viall 2008)	10
2.2	Contour of Temperature of Simulation Result (Viall 2008)	11
2.3	Pressure Test of Model (Viall 2008)	11
2.4	Temperature Isosurface (Viall 2008)	12
2.5	Temperature Output from CFAST (Viall 2008)	13
2.6	Experiment of Fire Test (Andersson and Lennqvist 2010)	14
2.7	FDS with Different Meshing Size and Simulation Result with (a) 20 cm Grid Size (b) 10 cm Grid Size (Andersson and Lennqvist 2010)	15
2.8	FDS Simulation with Embedded Mesh (McDermott et al., 2010)	16
2.9	Comparison of Simulation Results with Different Meshing Method (a) Single Mesh (b) Multi Mesh (Smardz 2006)	17
2.10	Thermal Stress Analysis Using ABAQUS (Benerjee et al., 2009)	19
2.11	Polygon Based Fire Visualization (Benerjee et al., 2009)	20
2.12	GPU Based Ray Traced Volumetric Rendering of Fire (Benerjee et al., 2009)	20
2.13	Fire Reconstruction Chronological Event (Yang, Xun and Wang 2011)	23

2.14	Comparison of Temperature between FLUENT and FDS at (a) 120 s (b) 360 s (Wang 2011)	24
2.15	Thermocouple Tree (Chai and Chow 2012)	27
3.1	Structural Modelling Using Revit Architecture	29
3.2	(a) Tetrahedron Mesh and (b) Cut-cell Mesh in FLUENT	31
3.3	Final Model of Fire Simulation in FLUENT	32
3.4	Model Display and Rectilinear Cell Mesh in PyroSim	35
3.5	(a) Single Meshing Method and (b) FDS Block View (Top View)	36
3.6	(a) Two Mesh Meshing Method with Coarse and Fine Mesh and (b) FDS Block (Top View)	37
3.7	(a) Embedded Fine Mesh into Coarse Mesh and (b) FDS Block (Top View)	38
3.8	Surrounded Meshing Method with Fine Mesh and 2 Coarse Meshes and FDS Block (Top View)	39
3.9	Revit Drawing of Full Scale House	45
4.1	Simulation Result of Fire Model Using Heat Flux as Fire Source (Plane Slice)	48
4.2	Temperature Isosurface at $t = 200$ s by FLUENT	49
4.3	Velocity Vector at $t = 200$ s by FLUENT (side view)	50
4.4	Simulation Result of Fire Model Using Non-Premixed Combustion (Plane Slice)	51
4.5	Simulation Result of Fire Model Using Non-Premixed Combustion with UDF at $T=30$ s (Plane Slice)	52
4.6	Comparison of Temperature of Non-premixed Combustion (a) without UDF and (b) with UDF (Volume Rendering)	52
4.7	Comparison of HRR Input for $D^*/\delta x$ Factor	55

4.8	Comparison of Temperature for $D^*/\delta x$ Factor at Height 2.7 m	55
4.9	Comparison of Temperature for $D^*/\delta x$ Factor at Height 0.6 m	56
4.10	Comparison of HRR under Different Meshing Method	59
4.11	Comparison of Temperature under Different Meshing Method at Height 2.7 m	59
4.12	Comparison of Temperature under Different Meshing Method at Height 0.6 m	60
4.13	Comparison of HRR under Different Meshing Method with FLUENT	62
4.14	Comparison of Temperature under Different Meshing Method with FLUENT at Height 2.7 m	62
4.15	Comparison of Temperature under Different Meshing Method with FLUENT at Height 0.6 m	63
4.16	Comparison of HRR between FDS and FLUENT	63
4.17	Comparison of Temperature between FDS and FLUENT at Height 2.7 m	64
4.18	Comparison of Temperature between FDS and FLUENT at Height 0.6 m	64
4.19	Hot and Cold Region Indication at Time = 300 s	65
4.20	Hot and Cold Region Indication at Time = 420 s	65
4.21	Hot and Cold Region Indication at Time = 540 s	66
4.22	Air Flow at Window by FDS	67
4.23	Comparison of Point Temperature between Meshing Method for Cylindrical Column	70
4.24	Comparison of Point Temperature between Meshing Method for Ramp	70
4.25	Comparison of Point Temperature between Meshing Method for Ramp (non-parallel)	71

4.26	Comparison of Point Temperature between Meshing Method for Hemisphere Structure	71
4.27	Plot3D Velocity View of Corner Fire	73
4.28	Velocity Comparison between Typical Fire (a) and Wall Fire (b) at Time = 4 s (plane slice)	73
4.29	Velocity Comparison between Typical Fire (a) and Wall Fire (b) at Time = 400 s (plane slice)	74
4.30	DNS Simulation Result of (a) Incident Heat Flux and (b) Fire Tracer	75
4.31	Comparison of Temperature Profile between (a) LES and (b) DNS Simulation Model	76
4.32	Comparison of HRR for LES and DNS	77
4.33	Source Code of Smoke and Fire Output (NIST 2012)	80
4.34	Interior FDS Block Display in SmokeView	81
4.35	Interior Actual CAD Model Display in SmokeView	81

LIST OF SYMBOLS / ABBREVIATIONS

ρ	density, kg/m ³
p	pressure, N/m ²
μ	dynamics viscosity, Ns/m ²
E	energy, J
T	temperature, °C
g	gravity acceleration, m/s ²
H	height of the domain
D	diameter of fire size, m
Q	heat release rate, kW
c_p	specific heat, kJ/kg.K
u	velocity component
K_H	heat transfer coefficient
f	vector of external force
N	number of grid points
L	length scale of the largest turbulent eddy
η	length scale of viscous effect
ϕ	property per unit mass
δt	timestep
δx	dimension of smallest mesh cell in x-axis
δy	dimension of smallest mesh cell in y-axis
δz	dimension of smallest mesh cell in z-axis
NIST	National Institute of Standard and Technology
FDS	fire dynamics simulator
GUI	graphic user interface

LES	large eddy simulation
DNS	direct numerical simulation
CFD	computational fluid dynamics
RANS	Reynolds-average Navier Stokes
RNG	renormalization group
HRR	heat release rate, kW
SDAA	smoke detector activation algorithm
CFL	Courant-Friedrichs-Lewy
SIMPLE	Semi-Implicit Method for Pressure-Linked Equations
PDE	partial differential equation
HRRPUA	heat release rate per unit area, kW/m ²
ACIS	Alan, Charles and Ian's System
DXF	drawing exchange format
UDF	user defined function
AMG	algorithm multigrid
ASCII	American Standard Code for Information Interchange
FORTTRAN	formula translating
CAD	computer-aided design
MPI	message passing interface

LIST OF APPENDICES

APPENDIX	TITLE	PAGE
A	UDF of Heat Flux Coding and Compilation Steps	92
B	UDF of Con-premixed Combustion Model	94
C	Analytical Step of Numerical Instability in FDS	96
D	Comparison of Temperature Slice, Velocity Vector Slice and Temperature Isosurface of Mesh Resolution Study	97
E	Comparison of Temperature Slice and Temperature Isosurface of Meshing Method	98
F	Comparison of Temperature Slice, Velocity Vector Slice and Temperature Isosurface between FDS and FLUENT	99
G	Comparison of Temperature Slice, Velocity Vector Slice and Temperature Isosurface of Various Geometry	100

CHAPTER 1

INTRODUCTION

1.1 Background

Traditional fire safety engineering and fire protection industry uses Standards and Codes in fire safety design in building and services field. In Malaysia, the “Guide to Fire Protection in Malaysia” is applied, which covers design concept, standards and codes, guidelines, charts, diagram and illustrations. These serve as a guidance for all fire industry practitioners to design the fire safety which is compliance to the Uniform Building By-Law.

The guide is published by The Institution of Fire Engineers (UK) Malaysia Branch with joint effort between Fire and Rescue Department Malaysia. The guide is divided into passive and active fire protection system. The passive protection system represents good planning, design and construction of a building while the active protection system represents the manual or automatic fire protection device such as fire alarm, smoke detector, heat detector and fire sprinkler, etc. Such guide is developed by a group of fire and safety engineering professionals. Therefore, the guide serve as a purpose for designing which the correctness of the design cannot be tested until a fire takes place. The flame behaviour and the smoke propagation cannot be illustrated and thus the actual fire situation cannot be predicted.

In modern fire safety engineering and fire protection, software is used to simulate fire through field modelling based on principles of computational fluid dynamics (CFD). The use of CFD includes dividing the domain or volume into mesh

cells and solving the equations of mass, energy and momentum conservation. The application of CFD in aerospace industry began during 1960s in the area of research and development, design and manufacture of aircraft and jet engines. The application of CFD further grew into internal combustion engine, combustion chambers of gas turbines and furnaces. In addition, CFD is also applied in the motor vehicle manufacturers to study the drag force, under bonnet air flow and in car environment. Since 1990s, the CFD has entered into a wider industry community due to its availability of affordable high performance computing hardware and the introduction of user-friendly interfaces.

As technology evolves and the increased fire protection concern using fire modelling software tools, the traditional approach of fire safety engineering and fire protection is now shifting towards modern approach. There are a few software currently available in the market such as Fire Dynamic Simulator (FDS), SmartFire and FireFOAM. In this paper, FDS will be used in the simulation, which imports the drawing drawn from Revit Architecture. Through data exchange package, .dxf format drawing is exported from Revit Architecture which is readable by PyroSim, which is a graphic user interface (GUI) of FDS.

Using FDS software, LES numerical method is applied to solve the low speed and heat driven flow of Reynolds-Averaged Navier Stoke (RANS) equations. The equation solving focuses on the smoke and heat transfer calculation and describes the turbulent fire combustion model. The output of FDS simulation allows users for post-processing includes the fire spread, flame behaviour, smoke propagation, temperature and HRR in SmokeView, which is included in the FDS package. The FDS is useful of displaying the interactive chronological fire event.

However, there is one major problem faced when using FDS. The FDS uses rectilinear mesh in the fire modelling simulation. From the past research, a coarse rectilinear mesh will results in poor in accuracy and poor meshing on curvature surface but the computational time is low. A fine mesh will result in good accuracy in simulation result and good meshing at curvature surface but the computational time is very long. It is necessary to find a suitable meshing method and size to optimize the simulation result and computational time. Therefore, in this research,

the study of the meshing method and size in FDS corresponding to the simulation result will be done. The possibility of FLUENT to simulate small compartment fire is also carried out and the simulation result will be compared with FDS. The simulation result from FLUENT will be used as a reference for comparison with FDS's simulation result in data verification.

Predictions of smoke flow patterns in buildings and in particular the design of smoke control systems are important elements of devising a fire safety strategy for a building. The need to control the flow of smoke within a building is to ensure the safe means of escape for the occupants of the building facilitate the fire service operations and protect the property.

In order to be able to select a suitable smoke control strategy, a fire safety practitioner needs to establish several parameters describing the dynamics of smoke flow in the compartments and the impact that smoke may have on the occupants of the building. These parameters typically include the temperature of the hot layer in the space of fire origin, the temperature of smoke in areas remote from the fire (e.g. in a smoke reservoir), velocities of smoke at selected locations, concentrations of combustion products, visibility, mass / volumetric flow rates of smoke, etc.

In predicting the smoke flow pattern in a building, fire safety engineers often refer to fire modelling tools such as simple empirical correlations, zone models or CFD models, also called field models. Therefore, there is a need in the fire engineering community for reliable fire modelling tools that can be confidently used to predict phenomena associated with smoke spread in buildings.

The development in fire design has an important role between authorities, clients, architects and fire safety engineers, mechanical and structural engineers in building projects. Cooperation among them is always important and the projects are intensified by the performance based design. The fire design provides a better fire safety of buildings, because the fire-related risks and scenarios have to be analysed carefully when using performance based fire design. In this coming research, the structural model will be drawn in Revit Architecture and the drawing is exported to PyroSim and FLUENT. A series of large eddy simulation (LES) fire model

simulations will be done in FDS and FLUENT. In FDS, the simulations with different meshing methods and sizes are carried out and compared with FLUENT's simulation result to study the accuracy of simulation result in curved geometry. The simulation results will also be attempted to be exported back into Revit Architecture for preview and walkthrough with fire and smoke. This is to create a linkage between structural modelling software and fire simulation software, which allows the architects to review the fire spread and smoke propagation based on their structural design.

1.2 Theory of CFD

In fluid dynamics, the motion of fluid can be described by a set of partial differential equations called the Navier-Stokes equations, where the three elements, mass, momentum and energy are conserved.

The equation of conservation of mass:

$$\frac{\partial \rho}{\partial t} + \nabla \cdot (\rho u) = 0 \quad (1.1)$$

where

ρ = density, kg/m³

u = velocity component

The equation of conservation of momentum:

$$\frac{\partial u}{\partial t} + (u \cdot \nabla)u = -\frac{1}{\rho} \nabla p - \nabla \phi + \frac{\mu}{\rho} \nabla^2 u \quad (1.2)$$

where

ρ = density, kg/m³

u = velocity component

p = pressure, N/m²

ϕ = property per unit mass

μ = dynamics viscosity, Ns/m²

The equation of conservation of energy:

$$\rho \left(\frac{\partial E}{\partial t} + u \cdot \nabla E \right) - \nabla \cdot (K_H \nabla T) + p \nabla \cdot u = 0 \quad (1.3)$$

where

ρ = density, kg/m³

u = velocity component

p = pressure, N/m²

E = energy, J

K_H = heat transfer coefficient

T = temperature, °C

The above equations are coupled differential equations, which is extremely difficult to find analytical solution. To find a solution, approximations of these equations have to be solved numerically using techniques such as finite element method, finite volume method and finite difference method. In CFD, approximations of the above equations are formulated and solved. One of the most important aspects of CFD modelling is the method of treating the turbulence. CFD models can be divided into three major groups: RANS models, LES models and Direct Numerical Simulation (DNS).

To understand the friction effects caused by turbulence, consider a control volume in a 2D turbulent shear flow parallel to the x-axis with a mean velocity gradient in the y-direction. The presence of vortical eddy motions creates strong mixing. Random currents that are associated with the passage of eddies near the boundaries of control volume transport fluid across its boundaries. These recirculating fluid motions follow the principles of conservation, which the mass cannot be created or destroyed, but fluid parcels transported by the eddies will carry the momentum and energy into and out of the control volume. The existence of the

velocity gradient, fluctuations with a negative y-velocity will generally bring fluid parcels with a higher x-momentum into the control volume across its top boundary and will also transport control volume fluid to a region of slower moving fluid across the bottom boundary.

The net result is momentum exchange due to convective transport by the eddies, which causes the faster moving fluid layers to be decelerated and the slower moving layers to be accelerated. Consequently, the fluid layers experience additional turbulent shear stress, which is known as Reynolds stresses. The shear stress contains second moments associated with correlations between different velocity components. However, the correlation between pairs of different velocity components due to the structure of the vertical eddies ensures that the turbulent shear stress is also non-zero and usually very large compared with the viscous stress in the turbulent flow.

Prior to the application of numerical methods, the Navier Stokes equation is time averaged. The momentum equation of the RANS model is as below:

$$\rho \bar{u}_j \frac{\partial \bar{u}_i}{\partial x_j} = \rho \bar{f}_i + \frac{\partial}{\partial x_j} \left[-\bar{\rho} \delta_{ij} + \mu \left(\frac{\partial \bar{u}_i}{\partial x_j} + \frac{\partial \bar{u}_j}{\partial x_i} \right) - \rho \overline{u'_i u'_j} \right] \quad (1.4)$$

where

ρ = density, kg/m³

u = velocity component

μ = dynamics viscosity, Ns/m²

f = vector of external force

Turbulence causes the appearance in the flow of eddies with wide range of length and time scales that interact in a dynamically complex way. The turbulence model for RANS focuses on the mean flow and the effect of turbulence on mean flow properties. The computing resources required for reasonably accurate flow computations are modest.

LES is an intermediate form of turbulence calculation which tracks the behaviour of the larger eddies. This method involves space filtering of the unsteady Navier Stokes equation prior to the computations, which passes the larger eddies and rejects the smaller eddies. The effect on the resolved flow due to the smaller, unresolved eddies is included by means of the so-called sub-grid scale model. Unsteady flow equations must be solved, so the demands on computing resources are high.

DNS computes the mean flow and all turbulent velocity fluctuations. The unsteady Navier Stokes equations are solved on spatial grids that are sufficiently fine to resolve the Kolmogorov length scale at which energy dissipation takes place and with time steps sufficiently small to resolve the period of the fastest fluctuations. DNS approach is highly costly and not yet of practical use in modelling fire due to the requirement for extremely well resolved mesh.

There are several RANS turbulent models available in commercial packages such as the k- ϵ model, eddy-viscosity model, mixing length model, RNG k- ϵ model, second-order closure model, Reynolds-stress model and near-wall model. In this research, both FLUENT and LES modelling in FDS will be explored for fire simulation.

1.3 Aims and Objectives

The aim of the research is to create a linkage between Revit Architecture and FDS. The structural model is drawn in the Revit Architecture and then export into FDS for simulation. The simulation output is then exported back into Revit Architecture so that the walkthrough in the Revit Architecture can be view together with the fire simulation. This is to show that the architects are able to view how the fire spread and smoke propagation based on the structural design.

Besides, this research is to determine a suitable way of meshing in the fire simulation using PyroSim with FDS integrated especially for complex geometry with

curved surface. The FDS meshes the model using rectilinear cell, which results in a coarse representation on the curved surface. Hence, it is important to use a suitable size of meshing before simulation is done. An over coarse mesh will result in poor representation of the fire model and poor accuracy but low computational power. On the other hand, an over fine mesh will result in good representation of the fire model and high accuracy but consumes high computational power. Therefore, it is important to use an optimal mesh size to balance the accuracy of the simulation with the computational power. With different meshes applied, the accuracy of the fire predictions made by FDS is determined.

With an appropriate mesh size chosen, the fire is simulated in FDS and then compared with FLUENT. This is done for data verification between FDS and FLUENT since no experimental data is available. The simulation is then done in a large scale model where a whole unit of house is simulated. The unit consists of more complex geometries and the meshing method used will have an effect on the curvature surface. The suitability of meshing methods to generate a stepped block for a curvature surface is also determined.

With the advancement of computer technology and the rise of fire safety awareness, this research will be able to provide a path for fire safety engineering to move into a modern approach to predict and simulate the behaviour of a real case fire scenario. This allows engineers to be able to develop an emergency response plan with the simulation result. This will significantly reduce the damages caused by a fire incident such as loss of lives and damage to properties.

CHAPTER 2

LITERATURE REVIEW

2.1 Fire Model Simulation Using FLUENT

In the research “Fire Modelling in FLUENT”, the author simulated a fire in FLUENT using two methods. These included the use of combustion model and k-epsilon turbulent flow model which coupled with energy equation and P1 radiation. The energy equation was turned on to simulate fire model using heat transfer to surrounding by conduction, convection, and radiation. The P1 radiation model was simulated to show the radiation of heat to the surroundings. The k-epsilon model is a two equations turbulent model that solves the turbulent kinetic energy, k and dissipation rate, ϵ in FLUENT. (Viall 2008)

The combustion model of a small residential compartment was used and divided two layers; an upper layer where the hot gases are trapped and a lower layer where cool air is trapped. The outlet vents at the window and door were divided into top and bottom to simulate the two way vent flow where hot air escape at top region of the vent while the cold air enters at the lower region shown in Figure 2.1.

The simulation result for non-premixed combustion model showed that this model was unsuitable for modelling a fire in a small compartment. The temperature shown was significantly higher than a real fire in the simulation. The temperature in the compartment can reach over 2000 K, which is much too hot for this type of fire as shown in Figure 2.2 below.

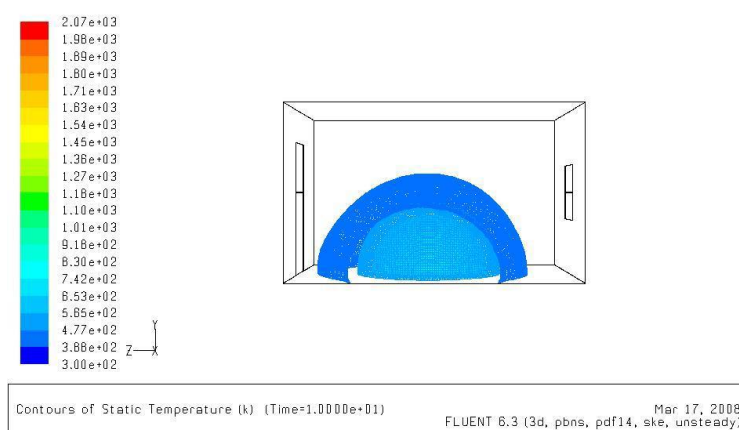


Figure 2.2: Contour of Temperature of Simulation Result (Viall 2008)

For the second fire simulation modelled as heat source with various boundary conditions, pressure boundaries were taken into considerations because pressure is involved in the dynamics of a fire scenario where the air flow and smoke are controlled. The result showed that there was no two way flow whereby the flow directions were at outward direction of the compartment at both openings at window and door after 120 s as shown in Figure 2.3 below. It was explained that a heat source model would not create hydrostatic pressure difference, which was unlikely a fire model simulated.



Figure 2.3: Pressure Test of Model (Viall 2008)

For the fire model using heat source, when the fire source's vents were set as velocity inlets and outlets, the temperatures reached very high, up to 500 K at the upper region of the room by the end of simulation. From the temperature isosurface as shown in Figure 2.4 below, it was determined that the upper hot region did not produce the result as expected. The author later compared the simulation result with CFAST for data verification.



Figure 2.4: Temperature Isosurface (Viall 2008)

In the CFAST, the fire model was run at 183 kW of fire, and it was found that temperature at upper layer of the room reached 150 °C (423 K), while at lower region, the temperature recorded at 31.5 °C (304.5 K) as shown in Figure 2.5. The CFAST model suggested that the FLUENT's fire simulation output was over predicting the temperatures.

The details of the vents in the fire simulation model as heat source in FLUENT were investigated and found out that heat was not released out of the vent in the compartment. The boundary conditions were adjusted and the P1 radiation model was used. The simulation was rerun and the temperatures shown in the upper layer was between 400 K and 420 K. These simulation results showed a closer results to CFAST fire model, however the lower layer temperature was inaccurate.

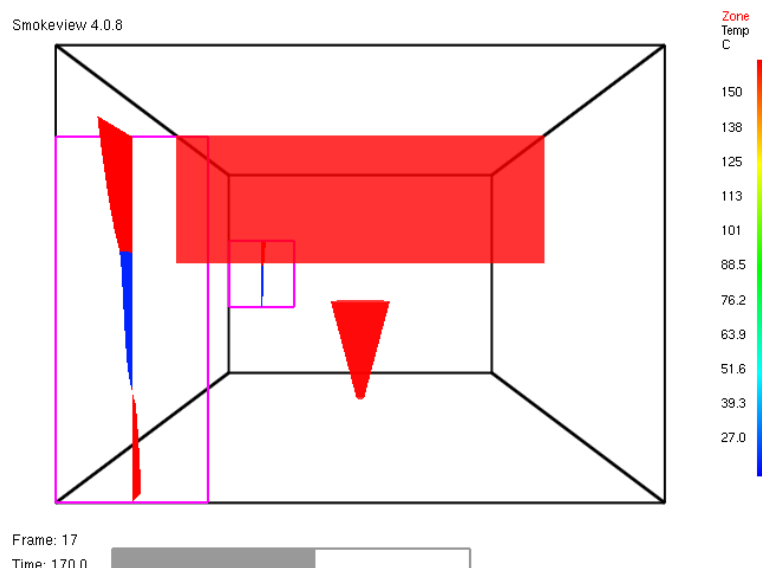


Figure 2.5: Temperature Output from CFAST (Viall 2008)

The author concluded that FLUENT's adiabatic non-premixed combustion model is inappropriate for modelling diffusion rate fire in a compartment. The author highlighted that FLUENT is capable to simulate fire to determine temperature in the far field but unreliable in near field data. It is understood that the adiabatic non-premixed combustion model are meant for simulation for furnaces, boilers, process heaters, gas turbines, and rocket engines. This is because this model involves the prediction on the flow field, mixing characteristics, temperature fields, species concentrations, and particulates and pollutants. The main problems encountered when simulating fire models in FLUENT are the fire size, boundary condition and data output.

The author had used other software to verify the results to ensure the correctness of the fire simulation. Hence, it is important to have different software to run the simulation for data verification. In this research, the FDS and FLUENT are used and the simulation results are compared with each other. The FLUENT is again used to determine the possibility to simulate the fire model. The latest ANSYS 14 with FLUENT will be used and the LES modelling is used instead of the k-epsilon model. The simulation done in the FLUENT will serve as purpose to verify the simulation result with FDS which will be done.

2.2 Fire Model Simulation Using FDS

FDS has been well-known for simulating structural fire model which is developed by NIST. A literature review is done to explore the various applications of the fire simulation using FDS done in other researchers.

2.2.1 FDS Simulation with Different Meshing Method and Mesh Resolution

In the research “A Burn-through Model for Textile Membranes in Buildings as A Tools in Performance Based Fire Safety Engineering”, the simulation of fire on textile membranes as a building material was done. Textile membranes are already on the market and used in a wide range of buildings, such as sports arenas, exhibition halls and other common rooms. (Andersson and Lennqvist 2010)

The authors had performed a literature study on the risks of using textile membranes as a building material. From this study the burning away of the membrane and the opening of a natural smoke evacuation were chosen for further studies. Fire tests of polyvinyl chloride / polyester membranes was performed in a ISO-room to examine their behaviour in fire and the burning away and hole opening in particular as shown in Figure 2.6 below. The authors had then simulated this experiment and explored the agreement between them and the FDS.



Figure 2.6: Experiment of Fire Test (Andersson and Lennqvist 2010)

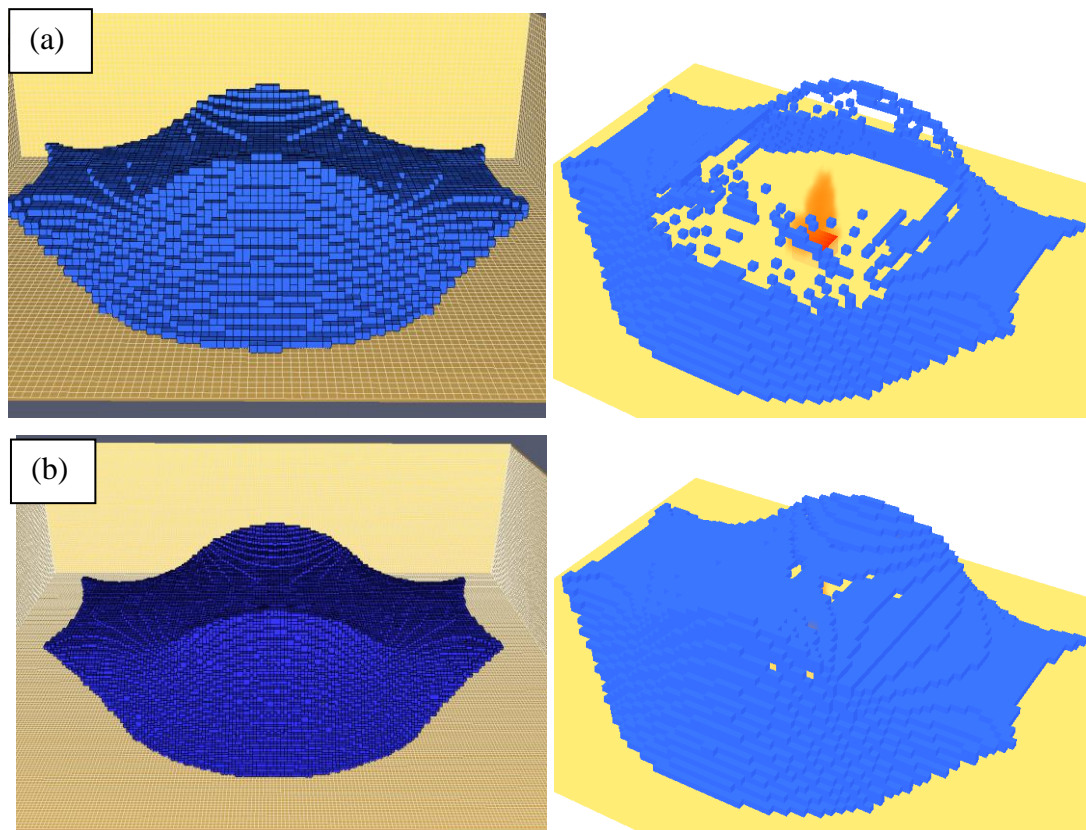


Figure 2.7: FDS with Different Meshing Size and Simulation Result with (a) 20 cm Grid Size (b) 10 cm Grid Size (Andersson and Lennqvist 2010)

From the simulation results, the author showed that it is possible to use the burn away option in FDS to model the opening up of a hole in textile membranes. However, no simulations have perfectly mimicked the validation experiments and the difference had been extensive in a lot of cases. The author also showed that with coarse grid used, the simulation result was not accurate compared with fine grid as shown in Figure 2.7 above.

The author concluded that opening of membrane would not occur until the flame reaches the ceiling. From the analysis of simulation result, the author highlighted that using textile membrane as a fire safety precaution for natural smoke evacuation was not reasonable. The report also showed that it is possible to import complex geometries to FDS and use the burn away option as well. However, only Cartesian coordinate inputs are allowed in FDS is a problem. This makes FDS impossible to accurately simulate complex geometries for fire safety design purposes.

However, with PyroSim, which is the GUI for FDS, the drawing complex geometry using the rectangular block is easier.

Since the import of complex geometry can be done to FDS, in this coming research, the structural geometry will be drawn in Revit Architecture and then export to PyroSim for simulation. The problem faced by this research is the accuracy in simulation due to the meshing size on the complex geometry. Thus, again, the mesh resolution study is an important factor in this research.

However, in the research “FDS: Complex Geometry, Embedded Meshes, and Quality Assessment”, a method for two-way coupling between coarse and fine meshes was outlined and demonstrated. For the problem considered in this work, the 1 m helium plume, a node reduction of more than a factor of four was realized for the embedded mesh case as shown in Figure 2.8 below. The method greatly simplified the problem of multi-resolution 3D mesh generation in FDS. From this research, it is possible to simulate the complex geometry which is faced in the research by Joel Anderson as discussed above. In this coming research, this method is used but the reduction factors of 2, 4 and 8 are tried on the curvature geometry only to enhance the meshing on the curved surface. This is to illustrate how the embedded mesh with different reduction factors can affect on the meshing as well as simulation output. (McDermott et al., 2010)

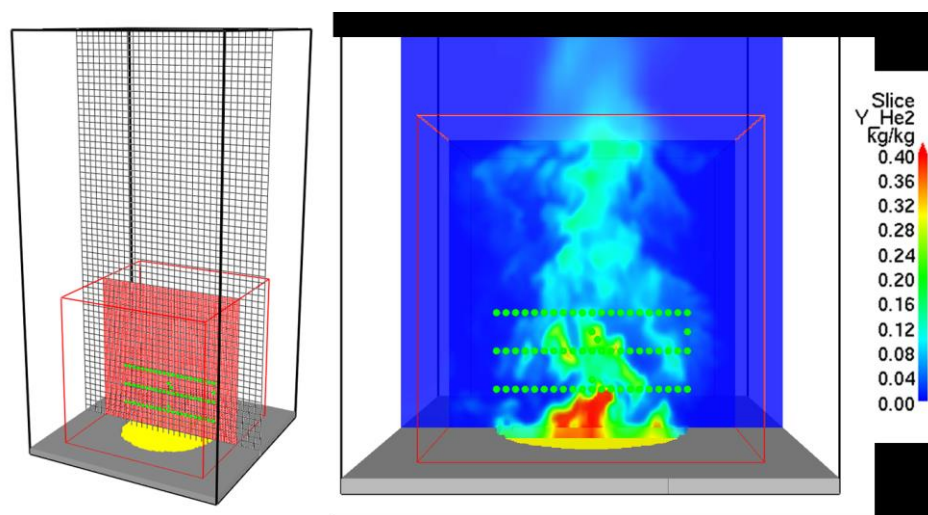


Figure 2.8: FDS Simulation with Embedded Mesh (McDermott et al., 2010)

The mesh sizing was further researched in the “Validation of FDS for Forced and Natural Convection Flow”. A series of coarse and fine mesh as well as burning fuel were simulated and compared with experimental result to investigate the influence mesh size in the simulation results. Some parameters were used to compare the result of each simulation with different mesh size such as burning rate, temperature, flow velocities, smoke flow pattern. For the fire and smoke behaviour, the comparison between simulation and experimental was based on visual observations. (Smardz 2006)

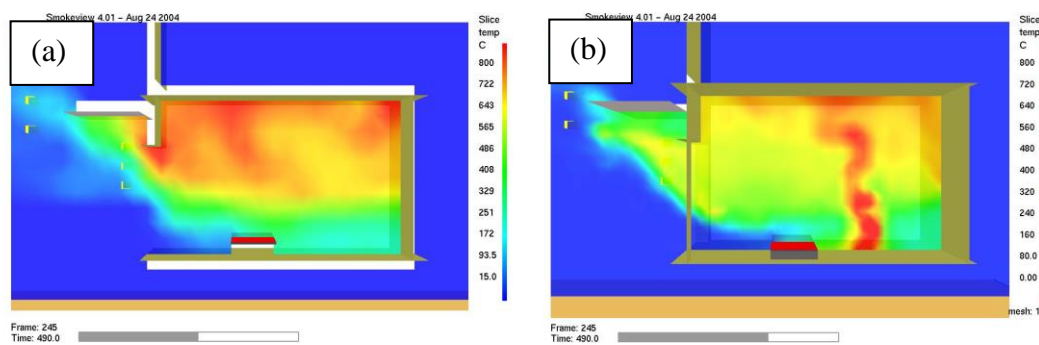


Figure 2.9: Comparison of Simulation Results with Different Meshing Method
(a) Single Mesh (b) Multi Mesh (Smardz 2006)

For temperature prediction, the accuracy of compartment temperature predictions depends strongly on the resolution of the computational mesh as shown in Figure 2.9 above. Generally, good predictions were only achieved where the size of the computational cells within the fire compartment was less than $1/10^{\text{th}}$ of the characteristic fire diameter. Where coarse mesh was used, the accuracy of maximum compartment temperature predictions was found to be poor, with temperatures consistently over-predicted by FDS. However, it is acknowledge that the difference between the experimental results and the FDS predictions might have been caused by the fact that thermocouples inside the compartment were covered with a thick layer of soot produced by burning the diesel oil.

The author highlighted that the mesh resolution would affect the HRR, which affected the overall simulation results. This shows the importance of selecting the

appropriate mesh resolution in fire simulation. In FDS, the mesh resolution is determined by the characteristic fire diameter. There were many researches on the mesh resolution to determine the suitable mesh resolution. As explained in the research by Piotr Smardz, the factor of characteristic fire diameter should be 10 in order to achieve accurate simulation results. This statement is further supported by the research done by N. Chai and W.K. Chow where the characteristic fire diameter factor should be more than 10. (Chai and Chow 2012)

However, with the characteristic fire diameter factor set to 10, the mesh cells produced can be up to millions of mesh cells in total. Based on the computation resources available, the simulation may not be able to be done. This limits the mesh resolution study in this research; however, the mesh resolution study is still carried out but at a lower characteristic fire diameter factor.

Therefore, the mesh sizing in the FDS is crucial in order to achieve high accuracy simulation result. In the coming research, the mesh sizing and method especially at the curved geometry are studied to achieve an accurate result in simulation. Furthermore, it is also vital to provide correct input for the HRR curve for fire simulation in FDS. Since in the PyroSim, the HRR ramp-up time can be entered and adjusted, the coming research will utilize this feature in the fire model simulation to achieve accurate simulation results.

2.2.2 FDS Simulation Linkage with other Software

In the research “Visualization of Structural Behaviour under Fire”, fire, thermal analysis of structures and deformation of structures were linked together. ABAQUS was used where the heat transfer by conduction is simulated to the steel beam for thermal analysis of structure. The output of the values of temperature at each nodal point from ABAQUS could be linked to the equivalent stress analysis in the thermal-stress analysis. The piecewise quadratic function was used to describe the temperature variation throughout the thickness. (Benerjee et al., 2009)

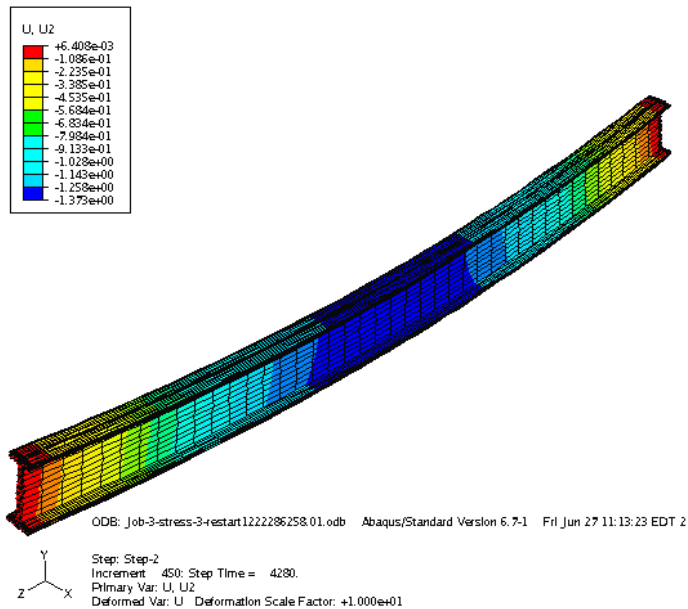


Figure 2.10: Thermal Stress Analysis Using ABAQUS (Benerjee et al., 2009)

In the deformation of structure, a gravity load was applied in ABAQUS as shown in Figure 2.10 to obtain structural deformation result. The finite element model developed for thermal analysis and structural deformation analysis model were linked to each other by the thermo physical and mechanical properties. During structural analysis, ABAQUS's restart feature was used where the finite element model of structural deformation due to gravity load was copied into a new model. Then, thermal load was applied based on the results file from the structural deformation analysis earlier. ABAQUS is able to interpolate data automatically between the thermal analysis and the structural analysis for the same geometry.

The outputs of the simulation were displayed in two visualization methods, which were polygon based visualization of fire (Figure 2.11) and GPU based ray traced volumetric visualization of fire (Figure 2.12). The author highlighted that the simulation could be done in a more complex structure with several structural members and connections. Furthermore, with the use of visualizations, the display of simulation result based on the thermal load could be enhanced.

In this research, the author successfully linked fire, structural deformation and thermal analysis using ABAQUS. The outputs can be preview in Polygon based

and GPU based for rendering. This shows that the linkage between ABAQUS and FDS can be done which enables the structural deformation due to fire possible to be simulated. However, in FDS, there is an extension where the simulation data from FDS can be directly translated into raw data. This is done using the `fds2ascii.exe` which converts unformatted binary data of FDS output to formatted text. With the formatted text output, the result from FDS at specific nodal point can be compared with ABAQUS result for verification. In this research, the FDS extension, `fds2ascii.exe` will also be explored to study if the simulation data from FDS can be extracted and exported into Revit Architecture.

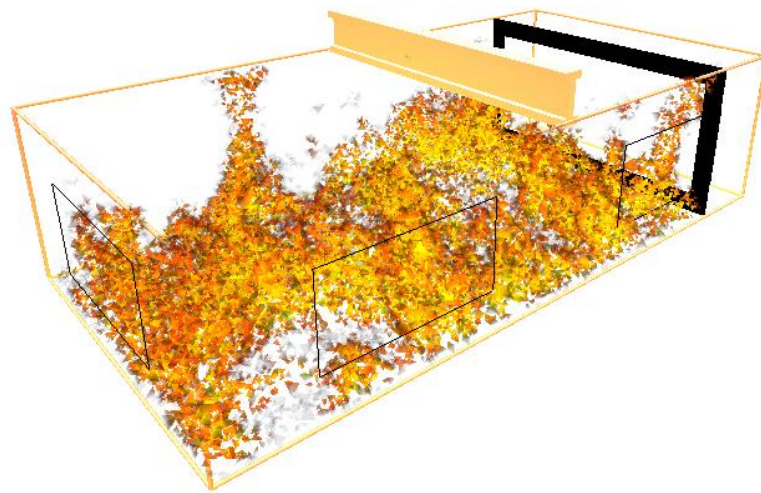


Figure 2.11: Polygon Based Fire Visualization (Benerjee et al., 2009)

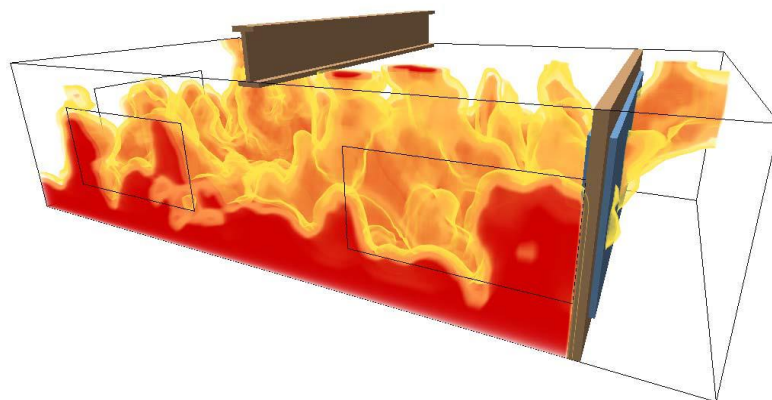


Figure 2.12: GPU Based Ray Traced Volumetric Rendering of Fire (Benerjee et al., 2009)

2.2.3 FDS Simulation with Other Fire Safety Devices

In the research “A smoke detector activation algorithm for large eddy simulation fire modelling”, using LES fire model, a smoke detector activation algorithm (SDAA) was developed and integrated in FDS. The SDAA described the response time of a smoke detector during a fire. The FDS ran the simulation to predict the smoke concentration and velocity next to the smoke detector. An algorithm based on characteristic velocity-based lag times was developed to describe the smoke flowing into the smoke detector’s sensing chamber. An experimental fire test was done in a multi-room compartment to compare with simulation results to determine the sensitivity of the smoke detector at different input parameters. It was found that at both high and low smoke velocity, the SDAA was very accurate in determining smoke detector activation time. (Olenick 2008)

For modelling the activation of sprinkler and heat detector in FDS, Runge-Kutta scheme was used. Two models were used in the simulation, namely Cleary model and Heskestad model. In the Cleary lag-time theory two components are included, which are mixing time and dwell time of a specific detector. On the other hand, the Heskestad theory only includes dwell time. To identify the difference in smoke entry characteristic time model, a wind tunnel was modelled over a range of velocities.

From the results, the characteristic of the smoke detector was determined by the filling time of the smoke into the sensing chamber. The filling time is strongly relying on the velocity next to the smoke detector. When the velocity was higher than 0.2 m/s, both Cleary and Heskestad predicted similar activation time. On the other hand, for velocities less than 0.1 m/s, the time to activate detector in Heskestad model was slightly slower compared to the Cleary, by tens to hundreds of seconds. Compared with experimental results, the SDAA algorithm provided good predictions on the smoke detector’s activation time in regardless of the velocity.

From this research, the author showed that it is possible to simulate fire model together with smoke detector. The heat detector, smoke detector and fire sprinkler are devices available in FDS. These devices can be added to the fire model

to simulate the effectiveness of the devices in operation at respective locations. This allows fire safety engineers to evaluate the effectiveness of the fire safety design. However, in this research, these devices are not used. The literature review of this topic shows the capabilities of FDS.

2.2.4 FDS Simulation for Fire Reconstruction

The simulation of fire model is also useful for fire accident reconstruction. The purpose of the fire accident reconstruction is to analyse the chronological event of the fire, thus allowing us to understand the cause of fire, how fire propagates and implements preventive measure. In the research “Experimental study and numerical simulation for a storehouse fire accident”, a full scale fire experiment was carried out on a shelf fire in a storehouse to study the fire ignition manner, fire spreading manner and fire burning characteristics. The experimental results was then compared with fire simulation done in FDS. The temperature, fire growth and heat release rate were compared between experimental and simulation data. The research showed another application of the FDS which is in fire reconstruction. (Yang, Xun and Wang 2011)

In both fire experiment and fire simulation, the fire went through three phases, which are initial growing phase, moderate growing phase and rapid growing phase. The fire growth and spread process in the simulation showed agreement with fire experiment as shown in Figure 2.13. However, the author highlighted that during experiment, one carton collapsed as the carton was burnt. This phenomenon is not able to be set in the FDS, which is a limitation of using fire simulation.

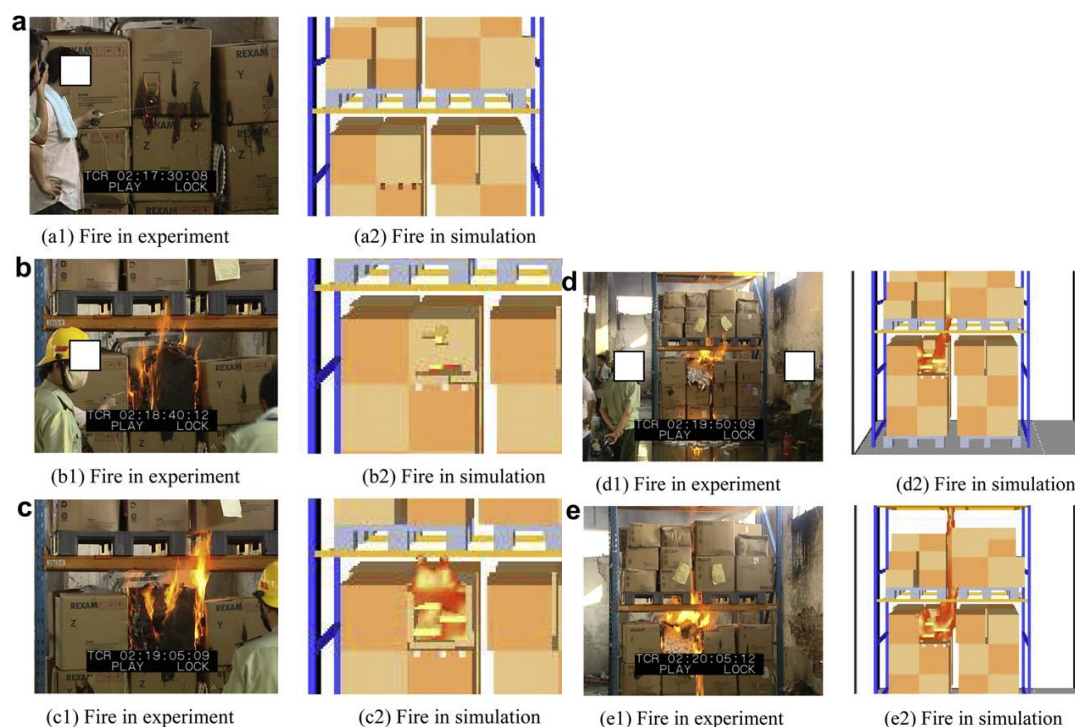


Figure 2.13: Fire Reconstruction Chronological Event (Yang, Xun and Wang 2011)

2.3 Comparison of Simulation Result between FLUENT and FDS

Since FLUENT and FDS are capable of simulating fire, there is a research to compare the output of the fire simulation from these two software. In the research “Comparative Research on FLUENT and FDS’s Numerical Simulation of Smoke Spread in Subway Platform Fire”, comparative study of their fire models was conducted in Tianjin Xiawafang Subway Station on the smoke spread and temperature. (Wang 2011)

In the simulation, the FLUENT has more options in simulation than the FDS, where FLUENT has eight turbulent models can be applied in various fields. However, FDS is specially designed for the fires simulation in the field of building. In addition, FLUENT has more advantages than the FDS in meshing because of the many meshing schemes available while FDS is only limited to rectilinear mesh. Furthermore, FLUENT is easier in modelling the geometry, at which a variety of

shapes are available in modelling. In FDS, the geometry can only be drawn by rectangular block while the meshing only limited to rectilinear cell. In post processing of simulation data, FLUENT has more analytical tools than the FDS.

The simulation results showed that in the middle of the train, in 120 s and 360 s, the simulation results from FDS were closer with measured results than from FLUENT's result in term of temperature as shown in Figure 2.14. Comparing the temperature curve between FLUENT and FDS, FLUENT's simulation result was much higher than FDS. In the FLUENT simulation, the fire source reached 5 MW in HRR but in FDS simulation, the HRR of fire did not reach 5 MW throughout simulation period.

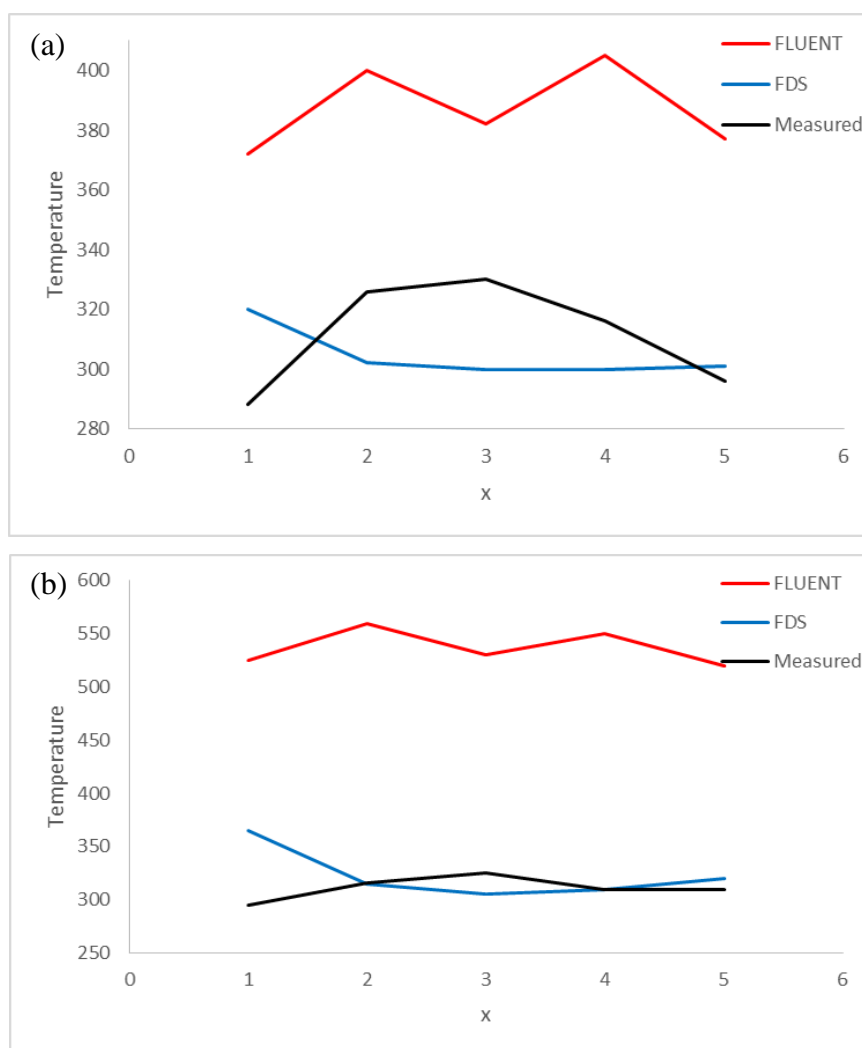


Figure 2.14: Comparison of Temperature between FLUENT and FDS at (a) 120 s (b) 360 s (Wang 2011)

The author highlighted that in order to achieve accurate result in both FDS and FLUENT, a mesh refinement should be done. From the comparison result between FLUENT and FDS, the FDS simulation result did not reach 5 MW of fire which causes the temperature to be lower compared with FLUENT. It could be noticed that from the simulation result shown in FDS, the fire grow was slower. As previously discussed in the Fire Modelling in FLUENT by Robert Viall, the fire simulation in near field using FLUENT is not appropriate but more suitable for far field modelling. The research done by Wang Binbin is in agreement with Robert Viall where the subway station can be considered as far field modelling. Using far field modelling in FLUENT, the author successfully simulated fire in this research.

It is important to understand how FLUENT and FDS solve the numerical simulation to understand how the solution is converged. There is difference approach in solving the numerical simulation in both FLUENT and FDS. In FDS, a non-interactive pressure solver, also known as Poisson solver is used to couple the pressure and velocity fields. It is not a SIMPLE-type algorithm. The timestep is calculated based on the Courant-Friedrichs-Lewy (CFL) condition. During calculation, the timestep is adjusted such that the CFL is within the acceptable range, which is ranged between 0.8 to 1.

The equation of the CFL condition:

$$\delta t = 5 \frac{(\delta x \delta y \delta z)^{\frac{1}{3}}}{g H^{\frac{1}{2}}} \quad (2.1)$$

where

δt = timestep in second

$\delta x, \delta y, \delta z$ = dimension of the smallest mesh cells

g = gravity acceleration

H = height of the domain

The CFL condition is a necessary but not sufficient to ensure the convergence of the solution. However, depending on the PDE and the scheme, the CFL condition maybe sufficient. FDS is second order accurate in space and time, therefore, the result can be obtained in higher accuracy with finer grid. However, it is impractical to use a very fine grid as it will have impact on the computational time. Therefore, it is required to use a suitable mesh size and method in the coming simulation. For the appropriate mesh size, it will be further discussed in the Chapter 3 Methodology on the characteristics fire diameter.

On the other hand, FLUENT uses the iterative method in solving numerical simulation by SIMPLE scheme. The FLUENT uses a fixed timestep and solves iteratively by monitoring the residual until the solution is converged. Usually, the solution is converged when the residual has reached a pre-set value or the number of iterations of each timestep. Comparing both methods used by FDS and FLUENT, they are using different methods in solving.

2.4 Post-Processing of FDS Simulation Results

In post-processing of the simulation, there are few methods to analyse the results. As suggested in the online FDS group discussion, there are few ways which are suitable for analysis as below:

- Energy balance check
- Mass balance check
- HRR check
- Velocity flow field
- Temperature
- Comparative results by experimental or other software package

In the coming research, the HRR check will be used to check if the input of fire source is as keyed in. Velocity of the flow field will be analysed as well in vector

form while temperature slice will be analyse the hot temperature layers. In the research Numerical Study on HRR in Room Fire on Liquid Fire under Different Ventilation Factor, the author had used a few thermocouples that line up to form a thermocouple tree to measure the point temperature at different height. This is to measure the temperature of each layer as shown in Figure 2.15. In this paper, this method will be adopted to measure the point temperature at different height to compare the point temperature (Chai and Chow 2012)

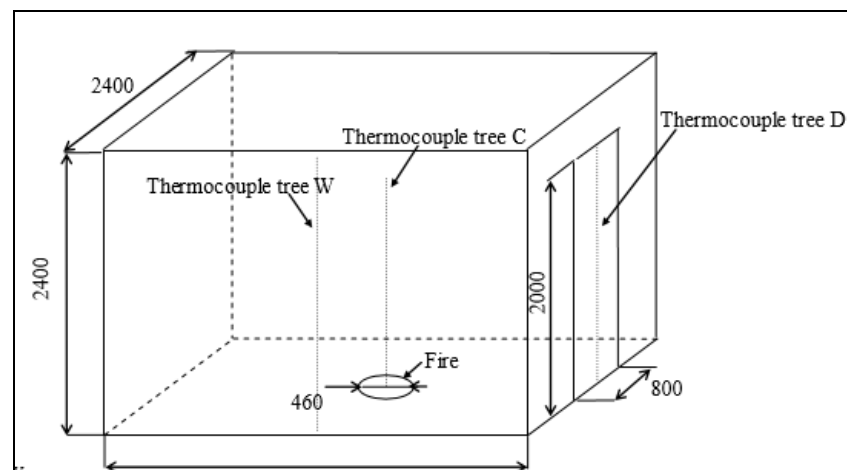


Figure 2.15: Thermocouple Tree (Chai and Chow 2012)

CHAPTER 3

METHODOLOGY

3.1 Structural Modelling Using Revit Architecture

The FDS study is started with the drawing of the structure of the building. The software Revit Architecture 2010 of Autodesk is used to draw the structural building. At the beginning of the research, a small room with dimension of 2 m length, 2.5 m width and 3 m height is drawn as shown in Figure 3.1. A simple curved wall is drawn at one corner of the room to study how the fire simulation software such as FLUENT and FDS do the meshing on the curvature surface.

Since the method of meshing used by both FLUENT and FDS is different, the simulation outputs are to be compared with each other. A roof, door, two windows and a sofa are included in the drawing. The sofa is placed so that fire simulation on the sofa can be done later, whereas the windows will be set as inlet or outlet ventilation. The roof is hidden so that in later fire simulation, the smoke propagation and fire behaviour can be view easily without blockage.

The reason of using Revit Architecture is that Revit Architecture is equipped with the real life walkthrough feature. Besides, the output of the drawing can be exported to FLUENT and FDS for fire simulation. It is also easier to draw a curvature geometry in Revit Architecture as the components and geometry modification feature are available in the software package. In contrast, FDS in PyroSim has more difficulty in drawing the curvature geometry as the geometry is mainly constructed by rectangular block.

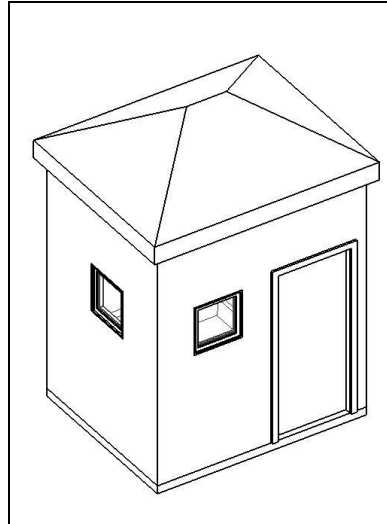


Figure 3.1: Structural Modelling Using Revit Architecture

For the data exportation to FLUENT, Alan, Charles and Ian's System (ACIS) is used where the drawing is exported as ".sat" file. On the other hand, the data exportation to PyroSim, the drawing is exported as Drawing Exchange Format (.dxf). The idea of this research is to link between Revit Architecture with fire simulation software, where drawings from Revit Architecture are exported to FLUENT and PyroSim for fire simulation. Then, from the simulation output, the result is exported back to Revit Architecture and to display the fire event in walkthrough mode. In this research, the possibility of exporting the simulation output to Revit Architecture is studied.

3.2 Fire Model Simulation in FLUENT

The FLUENT is integrated in the ANSYS; in the previous literature review, the research in the year 2008 by Robert Viall highlighted that the fire simulation in a small room is unsuitable as the temperature tends to get high very fast. In this research, the fire simulation is done again in FLUENT in the latest ANSYS 14 to determine the possibility of creating a diffusion rate fire in a small room.

Upon import the .sat drawing file from Revit Architecture, ANSYS's Geometry Modeller recognizes each side of the wall, windows, sofa, and door as individual components. This creates a long list of parts in the structure tree. A small square box is drawn on the sofa, which will be later defined as fire source. Besides, Geometry Modeller cannot recognize the air domain within the room, which means only solid parts, can be read by the Geometry Modeller. A primitive rectangular box is drawn over the structural building which will represent the air medium. Then, using Boolean operation, the overlapped region between the solid parts and the rectangular box is subtracted. This removes the overlapped region, leaving the air medium within the room. The model is then exported to Mesh in the ANSYS.

In the Mesh windows, there are two options of assembly meshing method offer in the ANSYS, which are tetrahedron and cut-cell mesh as shown in Figure 3.2. The tetrahedron mesh method divides the model and domain into triangular mesh whereas the cut-cell meshing method is the generation of hexahedral mesh with alignment with the Cartesian axis, which are the xyz axes. There are also other meshing scheme offered in ANSYS but other meshing schemes generate each mesh for each component or part of the structure without linking nodal. To avoid the program from meshing each solid parts and the air medium individually, the option "Assembly Meshing" must selected, else the mesh created will be on each individual parts with no relation to each other.

Under assembly meshing, two meshes are generated, which are tetrahedron and cutcell mesh. In meshing, cutcell, which consists of hexahedral mesh will provide a more accurate result as compared with tetrahedral mesh. However, the effort of generating the tetrahedron mesh is easier compared with cut-cell mesh, thus the time of resolving the equations is also faster. After generating the mesh, the tetrahedron mesh and cutcell mesh are export to FLUENT for simulation under ANSYS.

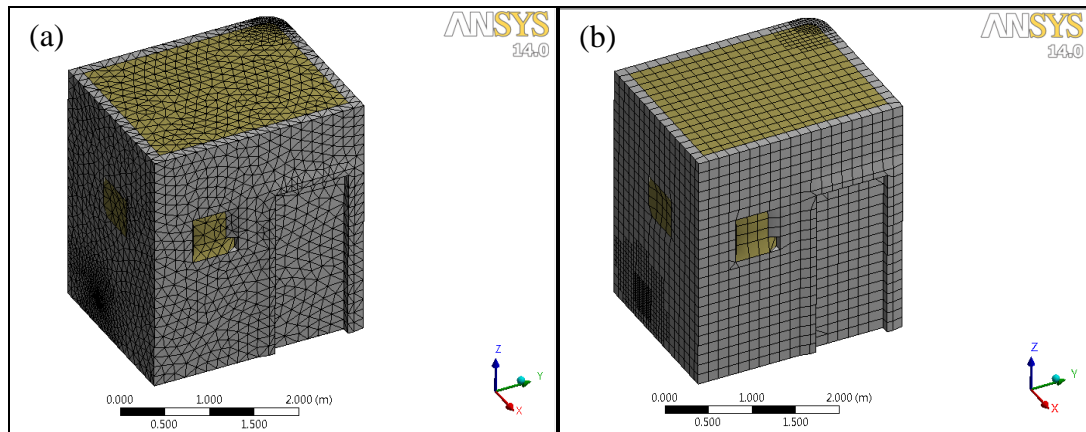


Figure 3.2: (a) Tetrahedron Mesh and (b) Cut-cell Mesh in FLUENT

In the FLUENT, the LES model is selected for fire simulation. There are two methods tried to simulate the fire. The first model is to mimic a fire by using heat flux while the second model is to use a non-premixed combustion model to simulate the fire. The fire model simulated in the FLUENT serves as purpose to verify the fire simulation's result in the FDS. To verify the result with the FDS, the HRR input and the temperature at specific time of the fire event at specific location especially at curved geometry will be compared. Besides, the vector flow field and temperature isosurface are also compared with FDS simulation result. For the initial research, the possibility of generating fire in FLUENT is studied first. A cross sectional plane at the fire source is created to view the fire shape, temperature from the two models simulated.

However, as the mesh is imported to the FLUENT, there is a problem with the boundary conditions. The boundary conditions are auto-assigned by the FLUENT. The boundary condition at the small square box should be recognized as "Wall" which can be later assigned as "velocity-inlet" but FLUENT auto-define it as "Interior". Under "Interior" option, the boundary condition, the "velocity-inlet" is not available to be chosen. With the boundary condition set as "Interior", the fire model is unable to be simulated although different boundary conditions under "Interior" group are tried such as "exhaust", "porous-jump", "wall".

The second try is by relocating the square box on the floor level. This is done by redrawing in the Geometry Modeller and re-meshing in the Meshing. Again, the

boundary condition for the square box which will be used as fire source is auto-assigned to “Interior”. It seems like FLUENT only recognizes the external layer of the whole meshing as “Wall” while everything internal the meshing is recognized as “Interior”

Therefore, for the third try, the floor level of the room is suppressed in Geometry Modeller, which means the floor level is hidden and excluded from re-mesh. The model is then re-meshed while maintaining the square box of fire source on the floor level. Upon read by the FLUENT, the square box (as circled in Figure 3.3) is recognized as “Wall”, which can be set as “velocity-inlet” for fire source.

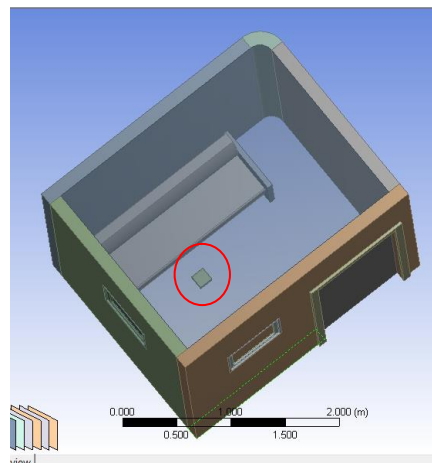


Figure 3.3: Final Model of Fire Simulation in FLUENT

However, there is another problem arise where the FLUENT groups several boundary condition together. In this case, the two windows, ceiling and floor level are group together as one boundary condition. The two windows are supposed to be set as “pressure outlet” to simulate smoke and pressure leak from the windows. Without the “pressure outlet” boundary condition, the simulation will goes divergence due to the pressure build up in the room; this will halt the fire simulation in FLUENT. The simulation is carried out although there is problem with the grouped boundary condition.

To solve this problem, in FLUENT, there is an option to separate the grouped boundary condition into individual boundary faces in the drop-down toolbar. With

the boundary condition's problem solved, the simulation is proceeded with the heat flux model and non-premixed combustion.

3.2.1 Heat Flux Model Using User Defined Function Input

Since the fire modelled consists of the ramp-up, steady and decay state, a user defined function (UDF) coding is generated. In the UDF coding, the value of the heat flux varies according to time. The UDF coding is generated using Notepad which the file is saved as ".c" and read by FLUENT. However, when compiling the UDF into the FLUENT, there is an issue with "NMAKE". The issue is due to the 64 bit version of ANSYS used, where the 64 bit version does not have a compiler. To solve the problem, Microsoft Visual Studio is installed in the computer. The detailed of the coding and compilation steps are explained in the Appendix A.

There are two heat flux models simulated where a normal heat flux model is simulated. This is done by just provide the UDF input in the heat flux parameter. For the second model, a Boussinesq heat flux model is used. The Boussinesq model is used for thermal buoyant flow. Since FDS solves thermal driven fire model, the Boussinesq model should be more suitable and compatible to FDS model. For Boussinesq model, the model can be turned on by just checking the Boussinesq operating density and temperature.

The fire simulation model is run to simulate a 15 minutes fire, which is same as FDS input, where a heat flux of 500 kW/m^2 is applied. For the post-processing, the data is exported to CFD-Post in the ANSYS package for further analysis.

3.2.2 Non-premixed Combustion Model Using User Defined Function Input

The purpose of simulating using non-premixed combustion is to generate a real fire where the chemical reaction is involved. In this non-premixed combustion model,

gasoline and air are used to generate fire with spark ignition. Unlike heat flux model, there is not heat release rate to be set. The only parameter which is able to alter the combustion is the velocity input and the fraction mean, which is the air fuel mixing ratio.

There are two attempts on simulating the non-premixed combustion model. The first model is just setting the velocity of the combustion inlet to 0.5 ms^{-1} and fraction mean of 0.5. For the second model, a UDF coding is applied to alter the velocity inlet profile for combustion. The velocity inlet of the fire source is modelled with a pyramid shape velocity profile which is shown in the coding in Appendix B. This is to simulate if the fire shape in real life where the air is drawn from side into the fire and form diffusion rate fire.

3.3 Fire Model LES Simulation in PyroSim

The PyroSim is equipped with FDS which is developed by NIST for fire modelling. From Revit Architecture, the drawing is exported as “.dxf” to PyroSim. In this research, the PyroSim 2010 and PyroSim 2012 with FDS 5.5.3 version will be used for fire simulation. In the FDS, the meshing of the model is only limited to rectilinear cell where the division is a rectangular box. Unlike FLUENT, there is no option for tetrahedron meshing. Using PyroSim, the structural model of the room is represented as obstruction. In addition, the FDS can recognize the interior of the room as air, which means the drawing of the air space as done in FLUENT is not required.

After meshing the model, the curved surface of the wall is represented as rectangular obstruction which adapts the curvature geometry as shown in Figure 3.4. With smaller mesh size, the smaller the rectangular mesh and thus better representation on curved geometry and better accuracy. Hence, in this research, different meshing methods are tried and simulated to compare the difference in the result and to determine the optimal solution for meshing so that the curved geometry of the structure can be well represented.

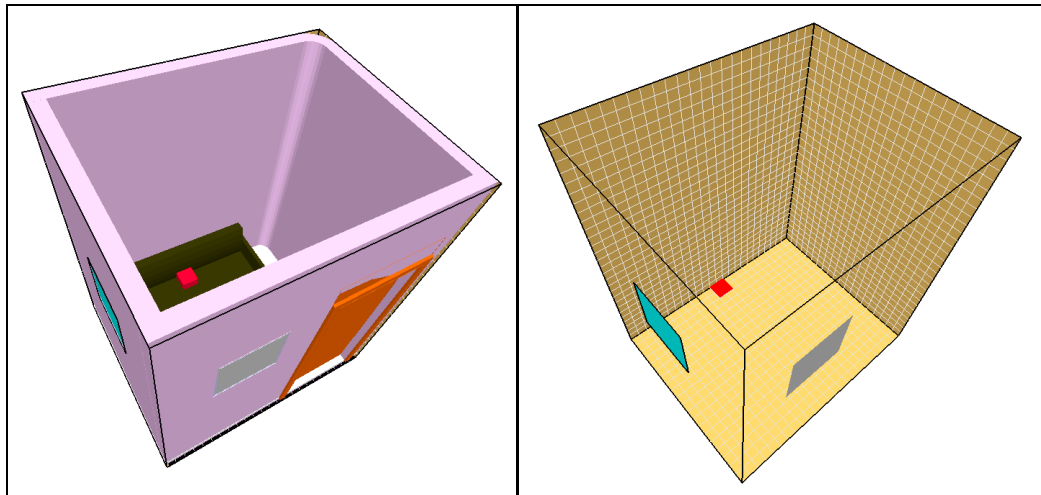


Figure 3.4: Model Display and Rectilinear Cell Mesh in PyroSim

3.3.1 Meshing Methods in FDS

There are five LES fire simulations using different meshing methods, which are:

- Single mesh
- Two mesh
- Embedded mesh with aligned mesh
- Embedded mesh with unaligned mesh
- Surrounded mesh

The different meshing methods are explored to study good representation on curved geometry without compromising the accuracy of the simulation result. To simulate the fire on the sofa, a small rectangular box is drawn over the sofa using coordinate system. On the top of the rectangular box, “vent” is drawn so that fire source can be defined. The fire rating is defined as Heat Release Rate per Unit Area (HRRPUA) in PyroSim. For each of the fire simulation under different meshing method, the HRRPUA is set to 500 kW/m² of fire for duration of 900 s. Besides, the total number of cells, computation time and simulation storage space for each sets of simulations are recorded for comparison.

The fire simulation model is done with the first meshing method with single mesh. The rectilinear meshing and FDS block are as shown in Figure 3.5 below.

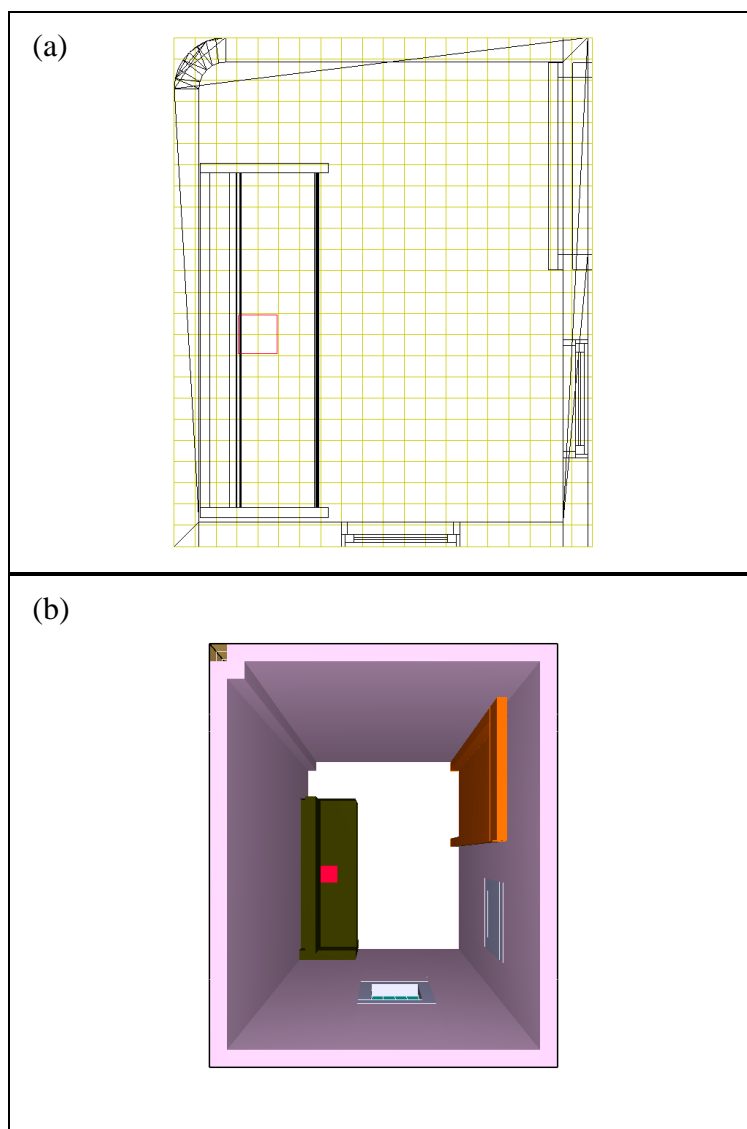


Figure 3.5: (a) Single Meshing Method and (b) FDS Block View (Top View)

The second meshing method is done using two meshing sizes on the model, where region with no curvature is meshed with coarse mesh while the region with curvature surface is meshed with fine mesh. The finer mesh is twice as smaller than the coarser mesh. It is important that the nodal point for both meshes to be in contact to prevent the misalignment error from the software. The combined mesh gives a total number of 26 100 cells. The rectilinear meshing and FDS block are as shown in Figure 3.6.

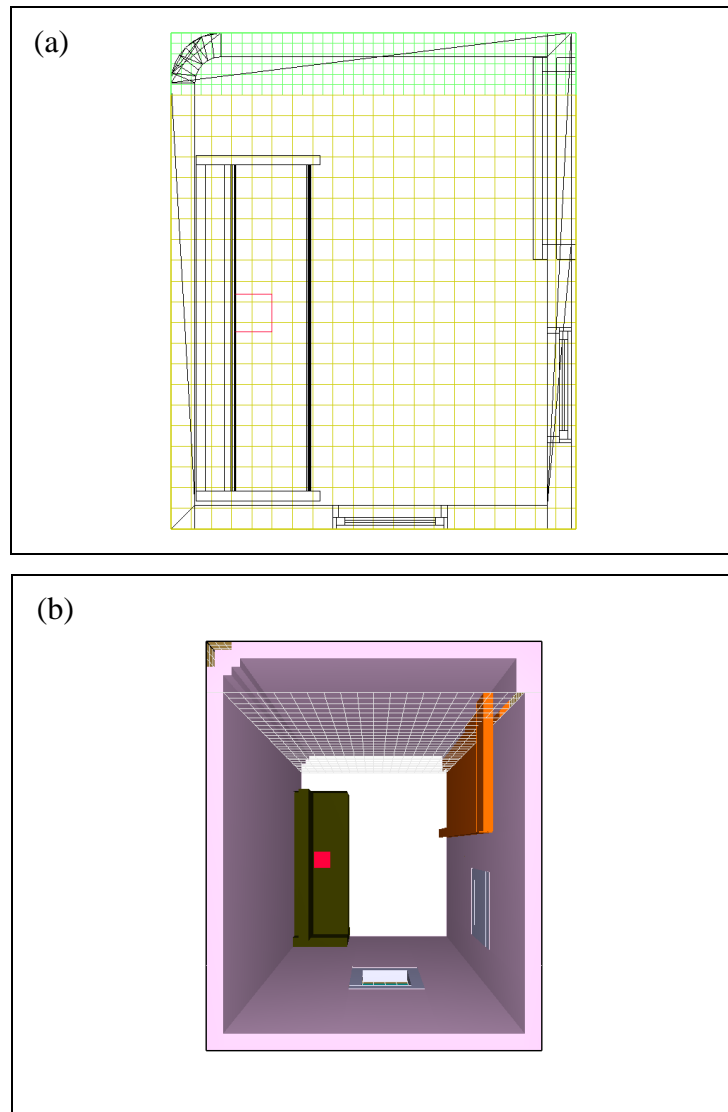


Figure 3.6: (a) Two Mesh Meshing Method with Coarse and Fine Mesh and (b) FDS Block (Top View)

The third meshing method is done by applying two different mesh sizes but the finer mesh is embedded into the coarse mesh at curvature surface. The first coarse mesh is applied onto the model while a second finer mesh, which is twice as smaller than the coarser mesh is embedded into the coarse mesh. It is important that the nodal point for both meshes to be in contact to prevent the misalignment error from the software. The rectilinear meshing and FDS block are as shown in Figure 3.7.

The use of embedded mesh with finer mesh serve as purpose to enhance the accuracy in a specific zone of a fire model. For example, if a fire simulation is more

focused on how a fire spread in a room, a finer mesh is embedded in a room in the primary mesh. However, in this research, the embedded mesh is applied to the curved geometry so that the coarse stepped curved geometry can be smoothen.

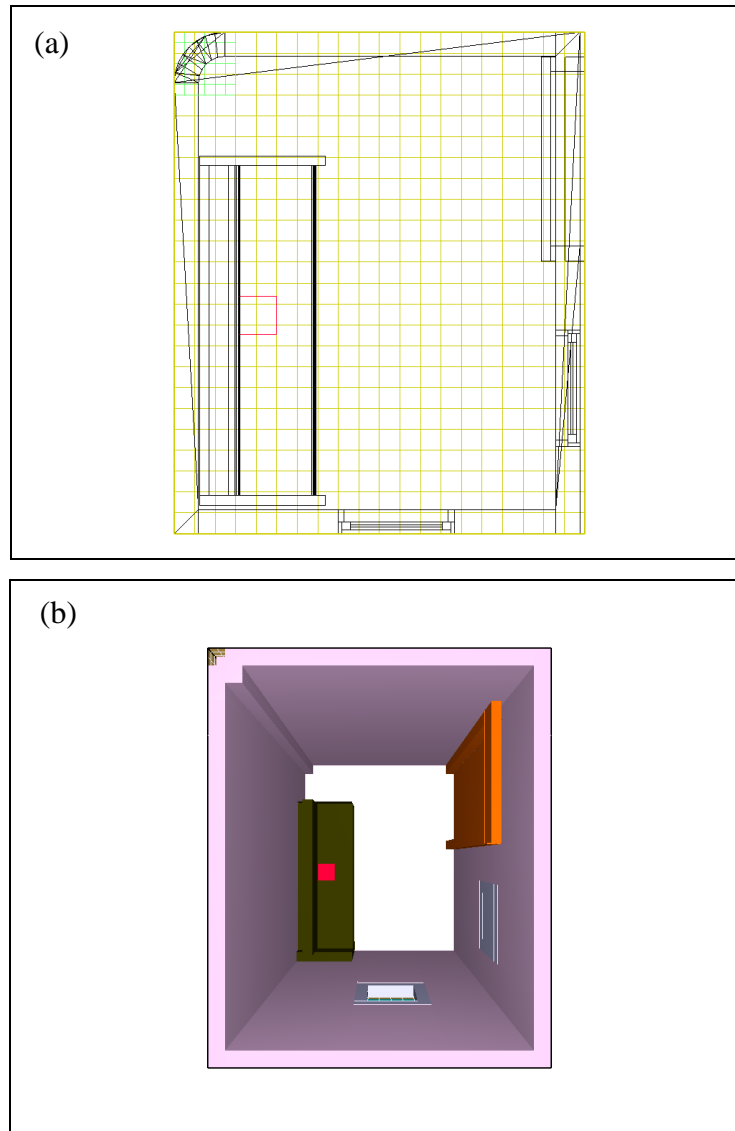


Figure 3.7: (a) Embedded Fine Mesh into Coarse Mesh and (b) FDS Block (Top View)

The fourth method is similar to the third model except the embedded mesh is not aligned to the primary mesh. As explained in the FDS user guide, the meshing must align so that the exchange of information in simulation can be done. It is not recommended to embed a mesh which is not aligned to the primary mesh. However,

in this research, this method is explored to study how the alignment of the embedded mesh will affect the simulation results.

Since meshing of PyroSim can only be done by entering the coordinates of the rectangle's diagonals, the fifth meshing method is done by using three meshes. The meshes are shown in the Figure 3.8 below, where the coarse meshes are applied to area without curvature geometry and the fine mesh is applied to the curved geometry of the wall. Although the nodal points at the boundary of each mesh are aligned with each other, the PyroSim 2010 gives a warning of misalignment in the meshing. This disallows the simulation from proceeding. The problem is solved when the PyroSim 2012 is used. In other words, the newer version of PyroSim 2012 had solved some program bugs of 2010 version.

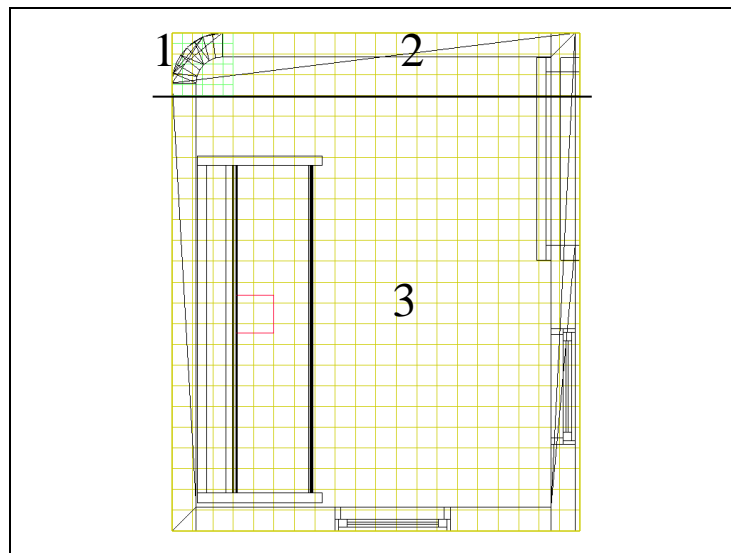


Figure 3.8: Surrounding Meshing Method with Fine Mesh and 2 Coarse Meshes and FDS Block (Top View)

3.3.2 Numerical Instability in FDS Simulations

Throughout the research, there are a few times where the simulation stopped due to the numerical instability. The numerical instability faced in this research is commonly due to the high velocity flow field. As explained earlier, FDS is designed

to solve low speed thermal driven flow. At such high velocity and in order to satisfy the CFL condition, the timestep is adjusted to very small, to the extent where the timestep is approaching zero. In this situation, the numerical instability occurs.

To check the cause of the numerical instability, the analytical steps are as explained in Appendix C. It is important to know some common causes of numerical instability so that these mistake can be avoided to ensure the successful run in simulation. There are few other possible causes which may lead to the numerical instability, which are:

- Aspect ratio of mesh cells are larger than 2:1. To solve this issue, the mesh cells should be constructed as close to a cube as possible.
- High speed flow through a small opening.
- A sudden change of heat release rate. The HRR input in FDS should not have large fluctuation or sudden rise or drop in HRR.
- A sudden changes to the flow field.
- Construction of mesh cells which does not fully cover the whole geometry. The mesh generation based on coordinate system should cover the entire geometry.
- Location of the fire source intercept at the mesh boundary between 2 mesh. The fire source should be located within a single mesh.

3.3.3 Appropriate Mesh Resolution

As explained in the Section 2.3 earlier, FDS simulation results are largely depending on the mesh size. The finer the grid size, the better the accuracy; however, the drawback of fine grid is the longer computational time and larger computational resources required. Therefore, it is required to identify an appropriate mesh resolution to ensure the balance between accuracy and the computational resources.

Commonly, the simulation is started with a coarse mesh and gradually refine the mesh until the simulation result shows no significant difference. This method is called the grid sensitivity study. However, due to the limited computational resource, it is not practical to use a very fine mesh size. Therefore, in FDS, there is an equation used to determine the suitable mesh size.

The equation of the characteristic fire diameter:

$$D^* = \left(\frac{Q}{\rho_{\infty} c_p T_{\infty} \sqrt{g}} \right)^{\frac{2}{5}} \quad (3.1)$$

or simplified

$$D^* = \left(\frac{Q}{1100} \right)^{\frac{2}{5}} \quad (3.2)$$

where

D^* = diameter of fire size

Q = heat release rate in kW

ρ_{∞} = density of air in kg/m³

c_p = specific heat in kJ/kg.K

T_{∞} = ambient temperature in K

g = gravity in m/s²

The ratio $D^*/\delta x$ is the number of computational cells spanning the characteristic diameter of the fire. In general, the more cells spanning on the fire diameter, the better the resolution of the calculation. Recall in Chapter 2 Literature Review, the characteristic fire diameter should be more than 10. However, based on previous research, a ratio of 4 to 16 in $D^*/\delta x$ is sufficient to ensure a good resolution. In this research, due to the limited computational resources, the ratio of 2, 4 and 8 are applied to study the appropriate mesh size on the curved wall model. The ratio is determined by adjusting the number of computational cells to achieve the mesh size,

δx . A few aspects are to be considered in selecting the suitable ratio such as computational time, computational resources and simulation results. (Hill et al., 2007)

3.3.4 FDS Simulation with Various Curved Geometry

Based on the simulation setups and results for each meshing method, which will be discussed in Section 4.2.2, the simulation sets with single mesh and embedded mesh are chosen for further simulation on various curved geometry. The purpose is to determine how the embedded meshing method is being constructed in each case. As discussed earlier where FDS uses rectangular block in representing the geometry, in curved geometry, FDS will represent in stepped block. This will create a rough step if the mesh size is too coarse. The application of embedded mesh on curved geometry will be able to reduce the coarseness of the stepped geometry, creating a smoother curved surface.

Also, from the simulation results which determine the appropriate mesh size in Section 4.2.1, the ratio of 4 for $D^*/\delta x$ is applied for all case. A few curved geometry cases had been set up to analyse how the embedded mesh is applied:

- Curved wall
- Inclined ramp
- Cylindrical column
- Hemispherical structure

In all case, the mesh generated are extended out of the geometry to how smoke travelled out of the windows. By default, the external boundary of the mesh are assigned as “Wall”. Therefore, at four sides of the external boundary, the boundary conditions are set to “Open”, which is specified as open surrounding. From all sets of simulation results, the HRR check, temperature slice, velocity flow field, point temperature and temperature isosurface are compared between single mesh and

embedded mesh to ensure there is no significant difference in the flow field in different meshing method.

A further study on the embedded meshing method is done where the size of the embedded mesh is determined as explained in the Literature Review on the reduction factor of embedded mesh. In all type of curved geometry simulation, the embedded mesh uses a mesh size twice as small as the primary mesh. In all case, the curved surfaces are well represented in smooth stepped surface. However, if given a very large geometry, a coarser primary mesh size will be used. Using the embedded mesh with twice as small as primary may not be sufficient to represent the curved geometry. Furthermore, with different size of the curved geometry, the embedded mesh may work differently.

Therefore, the study is carried out on the cylindrical column set where a different size of cylindrical columns are used, which are 6-inch, 12-inch and 24-inch diameter are used. In addition, the size of the embedded mesh if further extended into smaller size than the primary mesh, which are 2, 3, 4 and 8 reduction factor of the primary mesh size. The purpose is to find a relationship factor between the curved geometry, primary mesh size and embedded mesh size.

3.3.5 FDS Simulation with Various Type of Fire

During a fire, the location of fire in an enclosed volume plays an important role in the whole zone model. This is because in specifying the fire, HRR is used. The heat is released in a hemispherical area in a typical fire. However, when the fire is located just beside the wall, the hemispherical area is reduced by half. With the same amount of heat energy is released to half less area, the fire rises faster as compared to typical fire. The worst case scenario is when the fire is located at the corner where the heat is released to quarter of hemispherical area.

In the FDS simulation, these three type of fires are modelled, which are typical fire, wall fire and corner fire. With the same ramp input, same steady state of

HRR and decay state, the particular parameter, speed at specific time is analysed to compare the difference in the fire spread in each type of fire. The purpose of this simulation is to determine if FDS is able to simulate and mimic a fire based on different location and scenario.

3.4 Fire Model DNS Simulation in PyroSim

The PyroSim 2010 with FDS 5.5.3 is also equipped with options for simulation type, which are LES and DNS. As discussed in the theory in Chapter 1, LES is an intermediate form of turbulence calculation which filter the unsteady Navier-Stokes equation prior to the computations, which passes the larger eddies and rejects the smaller eddies. On the other hand, DNS computes the mean flow and all turbulent velocity fluctuations where the unsteady Navier-Stokes equations are solved; this requires high computation resources. In this simulation attempt, the DNS is tried to determine if the DNS type simulation is readily available to achieve a better fire simulation since DNS can solve the unsteady Navier-Stokes equations with smaller eddies.

In this research, the DNS simulation is tried by selecting the DNS in PyroSim to determine if the DNS is readily available to be used for fire modelling simulation by simply switching the simulation type from LES to DNS. For easy comparison, the first model, which is using a single coarse mesh is used. All other parameters like HRRPUA and simulation time remain the same.

3.5 Full Scale Model Simulation by FDS and Walkthrough in SmokeView

With the technical issues solved in the model of the small room earlier, the simulation is then done for a large scale model, which is a whole unit of house in FDS as shown in Figure 3.9. The full scale model is a more complex geometry as compared with the small room simulated earlier. With the large scale model

simulation, the overall picture of the simulation by FDS can be determined such as the rectilinear structure formed.

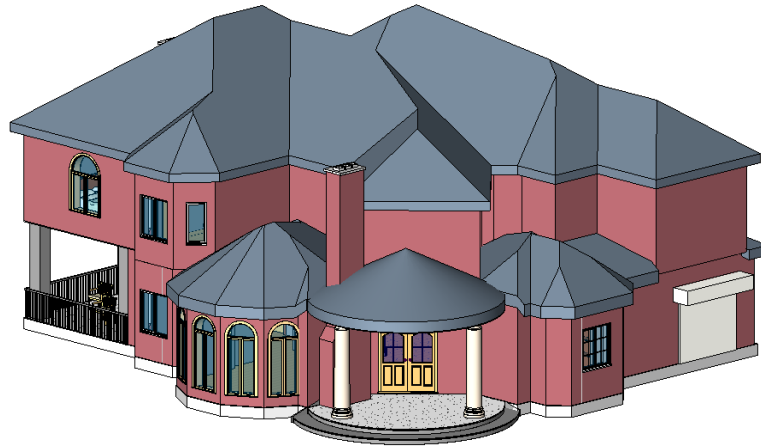


Figure 3.9: Revit Drawing of Full Scale House

When the rectilinear mesh is generated, the mesh size is adjusted to ensure the total mesh size is within the computational capability. A single mesh size of $0.15\text{ m} \times 0.15\text{ m} \times 0.15\text{ m}$ is applied to form 1.2 millions of mesh cells, which is within computational capability. To ensure the appropriate mesh size is applied, that is to obtain the fire characteristics diameter factor of 4, the fire size is set to 306 kW with fire span diameter of 0.6 m. The fire simulation is set for 900 s where 0 s – 300 s is the ramp up of HRR, 300 s – 600 s is the steady HRR and 600 s – 900 s is the decaying HRR.

After the simulation is done, the result can be view in SmokeView in the PyroSim. The SmokeView is the interactive windows which shows how the fire spread and smoke layer chronologically from the starting until the end of the simulation time. Besides, the simulation can also produce animated boundary quantities such as incident heat flux, soot particle, tracer particle, etc, which can be previewed in SmokeView. Such options should be checked before simulation start.

The walkthrough in the SmokeView is similar to the procedure as in Revit Architecture. In SmokeView, the walkthrough can be created by adding keypoints desired by entering the coordinates system at specific locations. The keypoints will

form a line, which is the path of motion of the real-life person walking in the structure. The view angle and time taken at each keypoint of the walkthrough can also be adjusted. In this research, a walkthrough is created with the fire simulation result and the possibility of exporting the simulation result back to Revit Architecture is determined.

CHAPTER 4

RESULTS AND DISCUSSIONS

4.1 Fire Model Simulation Using FLUENT

There are a total of 4 models of fire simulation done in the FLUENT, which are:

- Heat flux model
- Boussinesq heat flux model with UDF
- Non-premixed combustion model
- Non-premixed combustion model with UDF

4.1.1 Heat Flux Model

For the first model, which is simulated using heat flux as fire source from the small square box, there is no problem with the boundary condition despite the boundary condition is set to “Interior” by FLUENT. Under “Interior”, the boundary condition is changed to “radiator” which the peak heat flux of 500 kW/m^2 is entered.

However, from the simulation result, the output does not show a diffusion rate type fire. Instead, the output shows more likely a heat transfer from the square box to the surrounding. Based on the cross section area cut along the fire source as shown in Figure 4.1, the simulation output shows a heat transfer by radiation, which

is not the desired simulation output. The temperature near the square box reaches up to 306 K after a period of 10 s due to the low heat transfer coefficient of air. Another reason may be due to the boundary condition specified, where the “interior” or “radiator” boundary condition is not able to work correctly with the heat flux. In FLUENT, heat flux can be keyed-in in the “wall” boundary. This problem had been highlighted in Section 3.2 earlier.

From the simulation result, the fire model simulation using heat flux as fire source shows that the diffusion rate fire is not able to be done mainly due to the boundary condition specified. There is no desired fire shape formed in the simulation. Hence, the model simulated using heat flux as fire source is unsuitable to create the fire simulation.

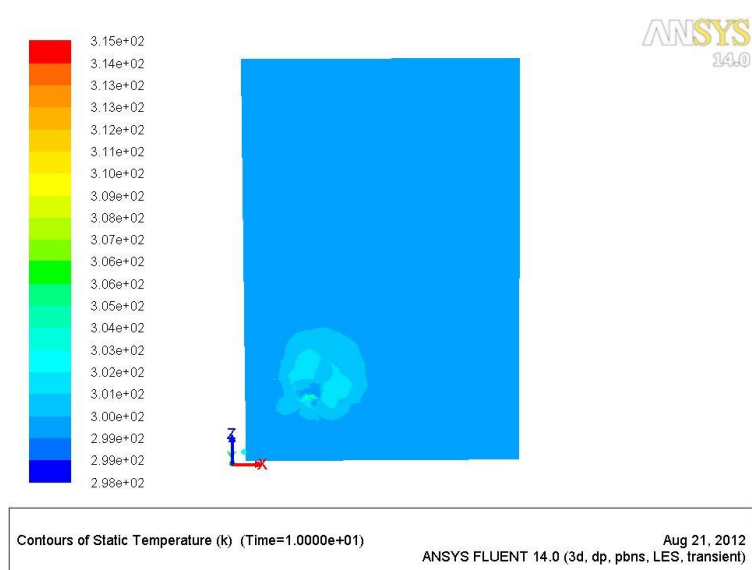


Figure 4.1: Simulation Result of Fire Model Using Heat Flux as Fire Source (Plane Slice)

4.1.2 Boussinesq Heat Flux Model with UDF

The Boussinesq heat flux model is used to solve problem related to buoyancy. In solving buoyancy problem, natural and mixed convection flows with gravitational force are considered. For the natural convection, the fluid flow is driven by the local

density variation while for the mixed convection flow, the convection fluid flow is driven by the pressure gradient and buoyant force.

The Boussinesq model uses the constant fluid density and local gravitational force which is then coupled with fluid thermal expansion coefficient in solving buoyancy problem. This creates a buoyancy reference temperature. From the temperature isosurface as shown in Figure 4.2 below, it can be shown that fire plume from thermal buoyancy is able to be created. The temperature rise also shown that the heat is release from the fire inlet. The temperature rise is more reasonable as compared with the previous heat flux model.

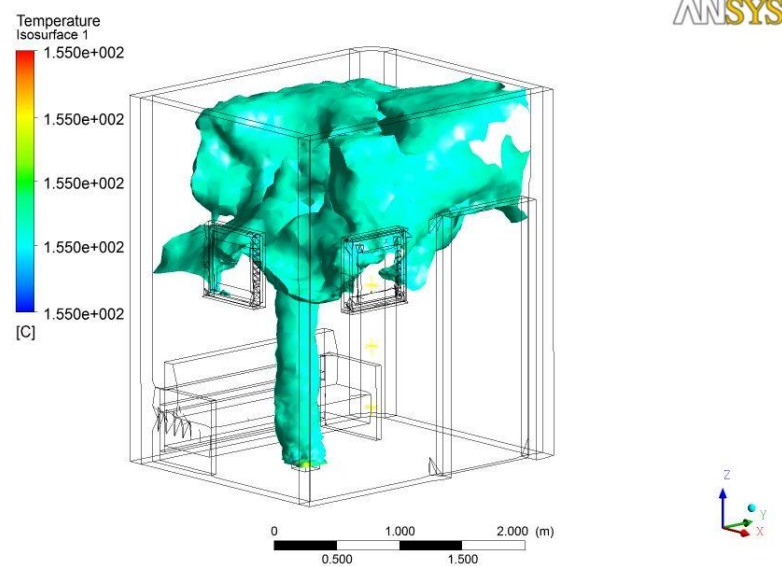


Figure 4.2: Temperature Isosurface at $t = 200$ s by FLUENT

The Figure 4.3 shows the velocity flow field at the window in term of vector. From the simulation result, it is shown that there is two-way flow at the window where the hot gas escapes at upper region while cold air from outside enters at the lower region of the window. As discussed in the Literature Review earlier, the researcher used the two-way flow vent to test the fire simulation. In this simulation model using Boussinesq heat flux model, the two-way flow is achieved. This proves that the thermal buoyant flow is successfully simulated as there is inflow and outflow of air due to the thermal difference.

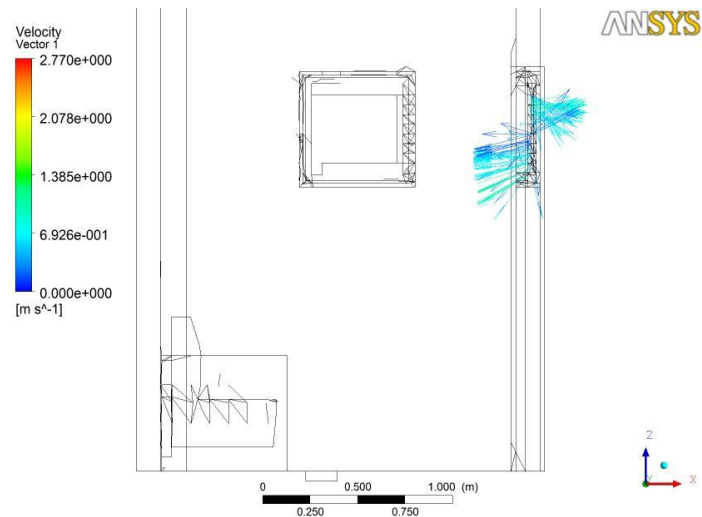


Figure 4.3: Velocity Vector at $t = 200$ s by FLUENT (side view)

From the simulation result using Boussinesq heat flux model, it shows a more appropriate fire model which is comparable to FDS. For this reason, the simulation result of FLUENT is later compared with FDS for data verification.

4.1.3 Non-premixed Combustion Model

For the fire simulation using non-premixed combustion model, a fire is successfully created in FLUENT using gasoline as fuel source. It is notice that the temperature reaches at maximum of about 1500 K at 30 s as shown from the plane slice at fire source as shown in Figure 4.4 below. The temperature rise is quite reasonable for a burning fire.

However, with the problem on the grouped boundary condition as discussed earlier in methodology, which is the windows, floor level and ceiling level are grouped as one boundary condition, the simulation result shows the fire go beyond the floor level and ceiling as shown in the temperature slice in the Figure 4.4. This problem is later solved by separating the grouped boundary condition which is explained in Section 3.2. However, the fire created seems like not a diffusion rate fire. Despite the air domain is defined as air, the fire mode created does not show

diffusion type fire. The fire created from the square box is more likely a jet stream of fire from the fire source inlet instead of a diffusion rate type fire.

From the ANSYS FLUENT user guide, the non-premixed combustion model is used for application like burner and not suitable for application of setting a fire based on HRR. Again, this non-premixed combustion model is inappropriate in simulating a fire in the room as the model used in the simulation is incompatible with the model in FDS.

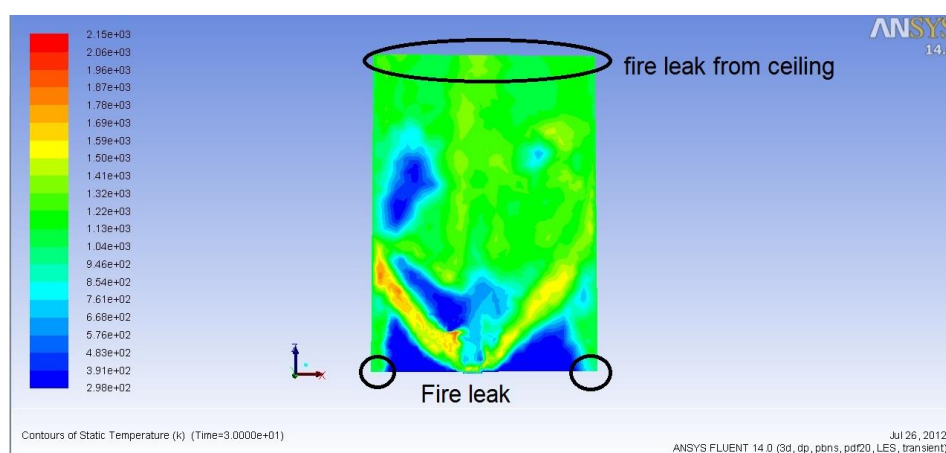


Figure 4.4: Simulation Result of Fire Model Using Non-Premixed Combustion (Plane Slice)

4.1.4 Non-premixed Combustion Model with UDF

Since the previous non-premixed combustion model could not simulate the diffusion rate type fire as desired. In this method, using the UDF, a velocity profile is created to simulate the fire in a pyramid shape, which is close to the diffusion rate fire shape.

Despite the velocity profile is defined as the pyramid shape, the outcome of the fire still shows the similar jet fire stream. From the Figure 4.5, it shows a jet fire with reasonably high temperature. The limitation of using the non-premixed combustion model is that there are too many unknown to be set in the input such as the fuel for burning, fraction mean mixture and fire inlet velocity. Although the UDF

is created to simulate the desired fire shape, the outcome does not show the desired results as shown in Figure 4.6. The limitation of using UDF in this case is that it is difficult to get the exact equation which is similar to fire in compiling the UDF.

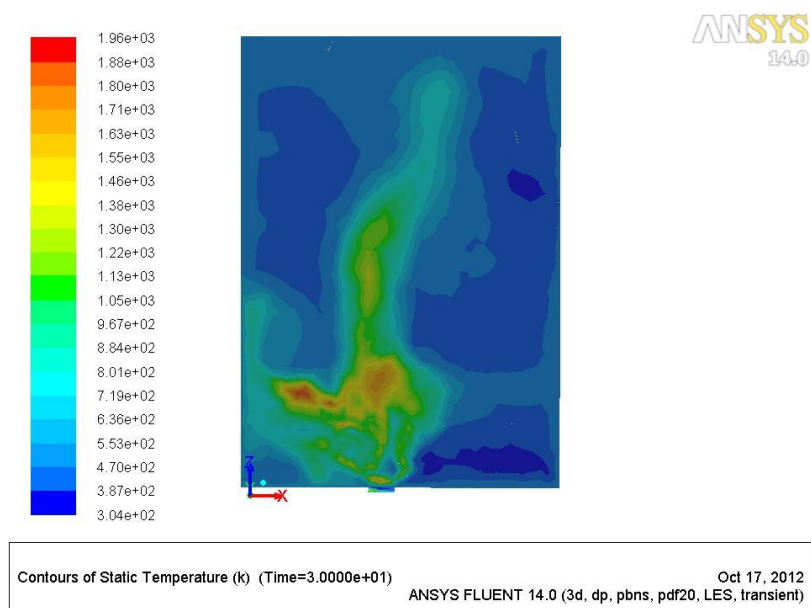


Figure 4.5: Simulation Result of Fire Model Using Non-Premixed Combustion with UDF at T=30s (Plane Slice)

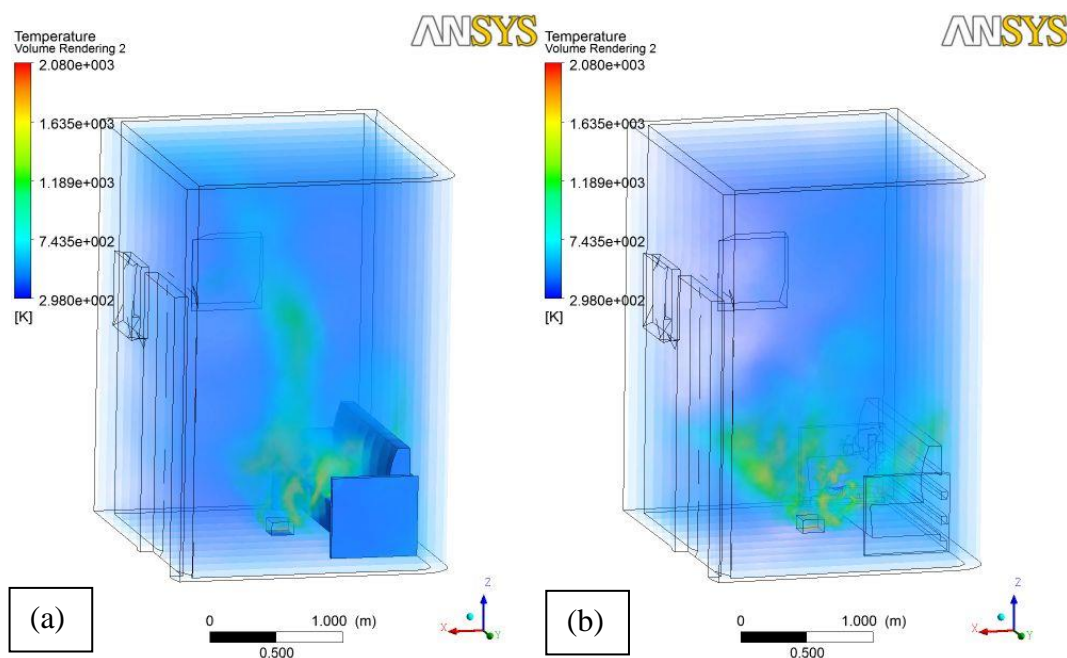


Figure 4.6: Comparison of Temperature of Non-premixed Combustion (a) without UDF and (b) with UDF (Volume Rendering)

For the meshing done in the Meshing under ANSYS, the meshing method used are tetrahedron mesh and cutcell mesh. Using the tetrahedral mesh, the effort in generating the mesh is easier than cutcell. From Figure 4.5 and Figure 4.6, the simulation is done on the tetrahedron mesh. However, when the cutcell mesh is simulated, the algorithm multigrid (AMG) problem occurs and halts the simulation. The problem is unable to be solved but it is suspected that it is the error during meshing. Using FLUENT, the simulation result can be exported to CFD-Post in ANSYS for post-processing. In CFD-Post, there are various options available for post-processing such as temperature, velocity, turbulence, density, etc. These are the advantages of using ANSYS over the FDS's SmokeView.

The main problem to simulate fire using FLUENT is the defining of the boundary condition. After the model is meshed, the FLUENT will automatically define the boundary condition for the model, which is either "Interior" or "Wall". For the external boundary of the mesh, FLUENT will auto define the boundary as "Wall" whereas the internal mesh is defined as "Interior". If the fire simulation is to be done in the room, the fire source will be automatically defined as "Interior", which in this case, cannot be set to velocity-inlet at the fire source to create fire. In the third try as describe in methodology, with suppression of the floor level, the fire source is actually located the external boundary of the mesh. With this condition, only the fire source can be set as "velocity-inlet" for boundary condition.

4.2 Fire Model LES Simulation Using PyroSim

For the fire model done in PyroSim with FDS, the mesh resolution study is done first followed with the meshing method. This is first to determine the appropriate mesh size to be used followed with the meshing method to be applied.

4.2.1 Appropriate Mesh Resolution

As discussed in the Chapter 3 Methodology on appropriate mesh resolution, the characteristic fire diameter serve as a guide to determine the appropriate mesh resolution. The ratio of $D^*/\delta x$ is used to determine the mesh size to be used based on the fire size and diameter.

From the past research, a ratio of 4 to 16 would be considered as good mesh resolution but preferably more than factor of 10. In this research, a ratio of 2, 4 and 8 are studied and the simulation results are compared with each set. The ratio is done until factor of 8 because further reduce the mesh size until factor of 16 will creates millions of cells. In this situation, the workstation is incapable of solving the simulation due to the high computational memory required. For comparison parameter, the mesh size, computational time, storage required and simulation results such as temperature slice, point temperature, HRR and isosurface temperature are compared. The Table 4.1 below shows a summarized result of the simulation with different ratio used.

Table 4.1: Comparison of Different Characteristic Fire Diameter Ratio

	Characteristic Fire Diameter Ratio, $D^*/\delta x$		
	2	4	8
Mesh Size	0.1 m × 0.1 m × 0.1 m	0.05 m × 0.05 m × 0.05 m	0.025 m × 0.025 m × 0.025 m
Storage	976 MB	6.83 GB	48.1 GB
Computational Time	1 hour 32 minutes	25 hours 57 minutes	317 hours 13 minutes

In the rules of thumb, as the mesh size is reduce by a factor of 2, the number of cells increased by 4 times while the computational time increased by a factor of 16 times. From the table above, the computational time obeys rules of thumb as the mesh size is reduced by factor of 2.

For the simulation results, these three sets of simulations are compared in the aspects of HRR, point temperature, temperature isosurface and velocity flow field. The purpose of the HRR check is to ensure the fire input source are the same with all sets of simulation. The HRR and point temperature of each sets of simulation are as shown below. Based on the HRR input graph in Figure 4.7, the HRR remains unchanged for all different ratio of $D^*/\delta x$. This shows that the ratio of $D^*/\delta x$ will not affect the HRR input.

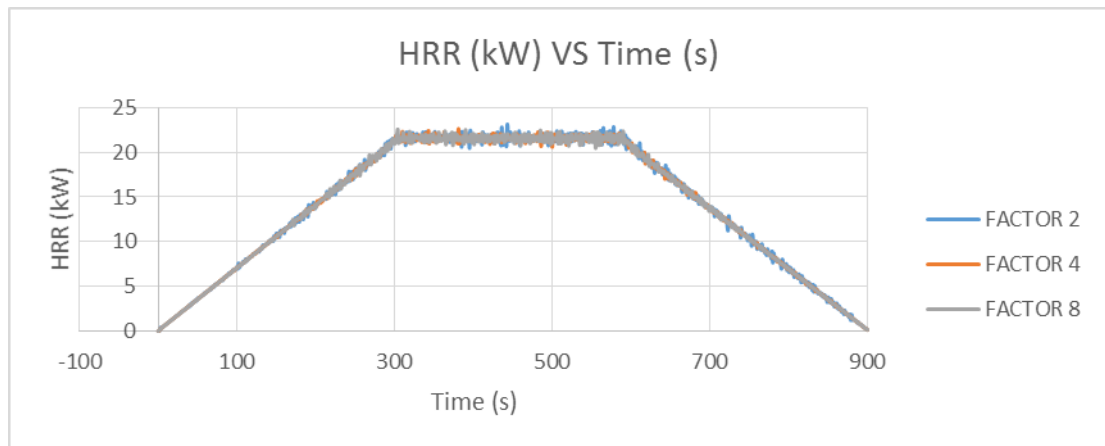


Figure 4.7: Comparison of HRR Input for $D^*/\delta x$ Factor

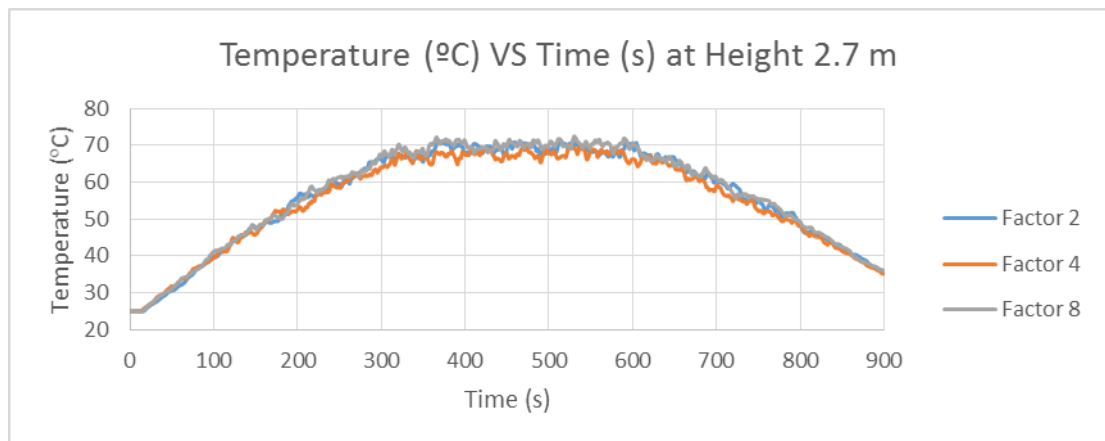


Figure 4.8: Comparison of Temperature for $D^*/\delta x$ Factor at Height 2.7 m

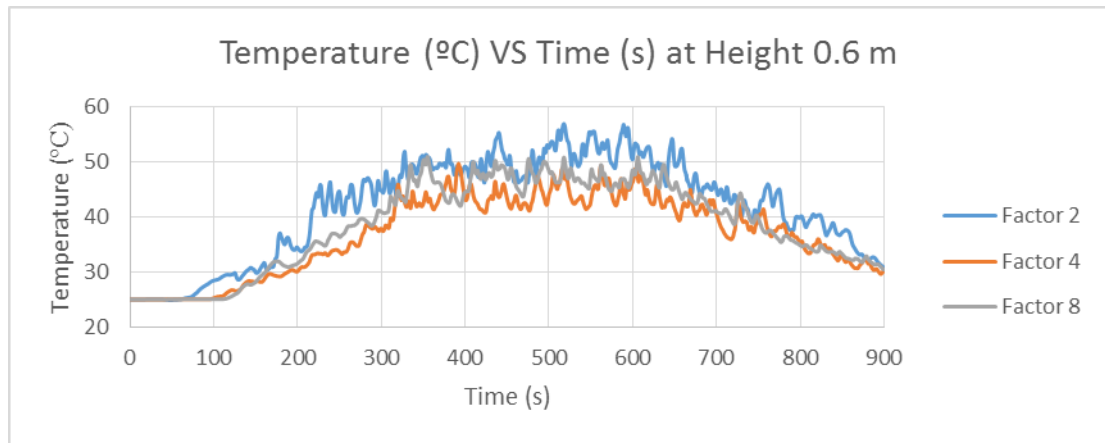


Figure 4.9: Comparison of Temperature for $D^*/\delta x$ Factor at Height 0.6 m

The thermocouples are fixed near the curvature wall to record the temperature layers as a method used by other researcher explained in Section 2.4. From the Figure 4.8 above, at the height of 2.7 m, there is not much significant difference in the point temperature among different characteristic fire diameter factor. However, at lower height at 0.6 m in Figure 4.9, it is shown that the point temperature for set (Factor 2) is higher than sets (Factor 4 and Factor 8). The point temperature shows more consistency for set (Factor 4 and Factor 8) as there is less variation in temperature.

For the comparison of the temperature slice, velocity flow field and temperature isosurface, the result is as per Appendix D, which shows the difference in temperature and velocity at quasi steady state where the HRR is constant. In this case, the period between 300 s to 600 s are compared with 100 s interval. This is because we are more interested at the result at the steady state of the fire, which is the steady state of HRR.

Comparing the temperature slice, velocity flow field and temperature isosurface, the results show difference in each sets. This is because from the HRR input curve, the curve shows fluctuation in HRR in all sets of simulation. By examining each specific time at interval of 100 s, it is not possible to obtain the similar results for each sets to be compared. The fluctuation in the HRR affects the velocity flow field in the room, thus generating slight variation in the flow field.

Hence, the velocity vector slice is not comparable. For the temperature slice, the temperature slice shows that set Factor 4 and Factor 8 are similar to each other while set Factor 2 shows more distinctive difference. To check if the velocity flow field have significant difference, the temperature isosurface of each sets are compared. All sets shows similar fire plume shape generated. Therefore, the variation of the velocity flow field can be considered as insignificant.

As previously stated, the desired factor for characteristic fire diameter should be in between 4 to 16 for deciding the mesh resolution. For a factor of 2, the point temperature is higher in prediction but the computational time is the fastest. For a factor of 8, the point temperature recorded is similar to factor of 4 but lower than set Factor 2. However, in set Factor 8, the computational time is much longer, which is undesired. Therefore, a mesh resolution with characteristic fire diameter ratio of 4 is more suitable as the computational time is reasonable without sacrificing the accuracy of the simulation result. For the rest of the meshing method simulation, a factor of 4 is applied.

4.2.2 Meshing Method in FDS

As discussed in the Methodology, there are 5 methods proposed in this research to determine the most appropriate way to mesh the geometry especially curved geometry to enhance the curvature stepped surface. The methods are as below:

- 1 mesh
- 2 mesh
- Embedded mesh with aligned mesh
- Embedded mesh with unaligned mesh
- Surrounded mesh

The Table 4.2 is the details of the simulation output under different meshing.

Table 4.2: Comparison of Meshing Methods in FDS

	Method 1	Method 2	Method 3	Method 4	Method 5
Meshing Method	1 Mesh	2 Mesh	Embedded Mesh with Aligned Mesh	Embedded Mesh with Unaligned Mesh	Surrounded Mesh
Computational Duration	21 hours 37 minutes	19 hours 21 minutes	22 hours 21 minutes	36 hours 32 minutes	25 hours 58 minutes
Simulation Storage Space	6.23 GB	4.27 GB	6.56 GB	6.56 GB	6.83 GB
Numerical Instability	No	Yes	No	No	No

From the Table 4.2 above, the use of Method 3 (Embedded mesh) has little impact on the computational time. This is because the total number of mesh cells is slightly higher than the Method 1 (1 mesh), thus requires slightly more time in simulation.. As compared with Method 2 (2 mesh), there are more mesh cells generated and hence have more impact on the computational time. It is also noted that the Method 2 (2 mesh) encountered numerical instability which stopped halfway in the simulation. The computational duration for Method 2 will be a lot longer than any other method if there is no numerical instability due to the large number of mesh cells as compared with other method. Even precautions have been taken to avoid numerical instability as mentioned in Section 3.32, the numerical instability still occurs as in Method 2. The numerical instability happened in Method 2 may be due to the problem in the exchange of information between 2 mesh boundaries even though the nodal points between 2 meshes are aligned together. This highlights the problem that may be faced when using more than 1 mesh in simulation. However, with all meshing methods used, there is little impact on the computational storage. Hence, the computational storage is not taken into consideration in method selection but the computational time is considered.

For comparison on the simulation results under different meshing methods, the HRR and the point temperature at the curved wall area are compared. A thermocouple tree is fixed near the curvature wall to record the temperature layer.

Besides, the temperature slice at the curvature wall and temperature isosurface are also compared among all meshing method, which is shown in Appendix E. The point temperature at the height of 2.7 m and 0.6 m are shown to determine how the meshing method affects the temperature at high and low level of the building. The HRR and point temperature comparison are as shown in Figure 4.10 to Figure 4.12 below.

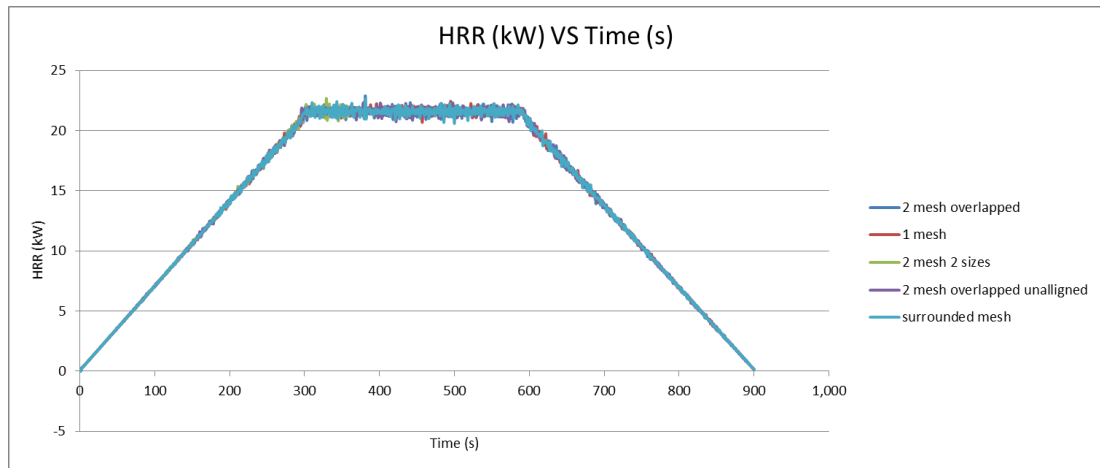


Figure 4.10: Comparison of HRR under Different Meshing Method

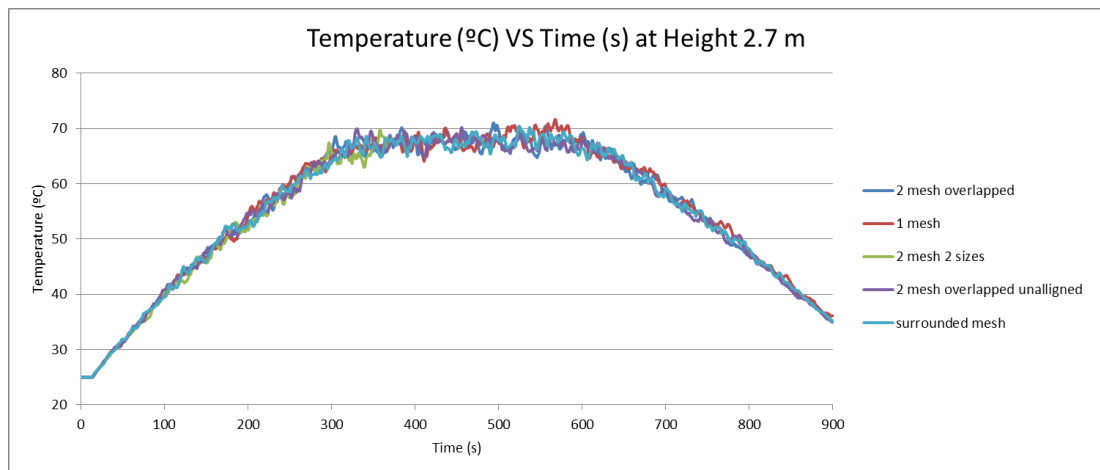


Figure 4.11: Comparison of Temperature under Different Meshing Method at Height 2.7 m

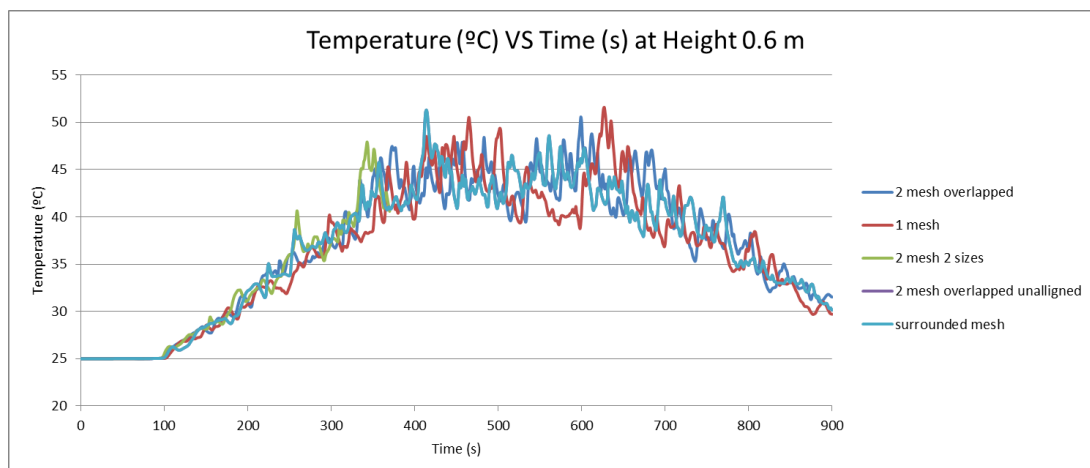


Figure 4.12: Comparison of Temperature under Different Meshing Method at Height 0.6 m

From the Figure 4.10, all meshing methods show that there is no affect on the HRR input while the point temperature in Figure 4.11 and 4.12 show consistency in both height level of the building at 2.7 m and 0.6 m for all methods of meshing. There is no significant difference in the temperature where there is no sudden spike of temperature rise or drop. The variation of temperature for all meshing method is less than 10 °C, which is considered small in a fire scenario. Hence, the variation in temperature difference can be neglected. From the simulation results and output from all meshing method, a matrix table is created to show the suitability of the meshing method with a few criterions. This matrix table will decide the meshing method to be used for further simulation on complex geometry.

From the Table 4.3, Method 1 (1 mesh) and Method 3 (Embedded mesh with aligned mesh) are the most appropriate methods to be used in meshing the complex geometry based on the score point. With the meshing method selected, this embedded mesh method is further implemented together with the 1 mesh method in further research on the various type of curved geometry.

Table 4.3: Decision Matrix of Meshing Method Selection

	Method 1	Method 2	Method 3	Method 4	Method 5
Meshing Method	1 Mesh	2 Mesh	Embedded Mesh with Aligned Mesh	Embedded Mesh with Unaligned Mesh	Surrounded Mesh
Computational Time	+	-	+	-	E
Ease of Mesh Setup	+	E	E	E	-
Tendency of Numerical Instability	+	-	+	+	+
Stepped Surface Quality	-	+	+	+	+
Total	3+, 1-	1+, 1E, 2-	3+, 1E + Good	2+, 1E, 1- E Normal	2+, 1E, 1- - Bad

4.2.3 Comparison of Simulation Results with FLUENT and FDS

From the simulation results in Section 4.1.1, only Boussinesq heat flux model in FLUENT is most compatible and most suitable to be used to compare the results with the FDS. The purpose of the comparison is for data verification of the simulation results. In data validation and verification, a real fire experiment should be done so that the exact results can be obtained. However, due to the limited resources, the fire experiment is not carried out.

Based on Figure 4.13, the HRR shown in the FLUENT is higher than the FDS. The FLUENT recorded a peak HRR of 29 kW while the FDS recorded a peak HRR of 22 kW. Note that the FLUENT data is incomplete due to the power trip during simulation. However, the simulated result is sufficient for analysis as the interest is at the steady state of HRR. The difference is due to the area of the fire source inlet drawn in the FLUENT is larger than FDS. With larger fire source area,

the fire input heat flux per unit area in FLUENT will result in higher HRR. In addition, from the point temperature graph in Figure 4.14 and 4.15, it is also shown that the point temperature in FDS is much lower than the FLUENT. It is found that the temperature predicted by FDS is lower than expected in a fire scenario. When the input of FDS is checked, the option “Convective Heat Flux” in the FDS is not turned on. Without this option turned on, the fire simulated using HRRPUA gives a natural convection of heat transfer. On the other hand, the convective heat flux gives a forced convection heat flow in simulation.

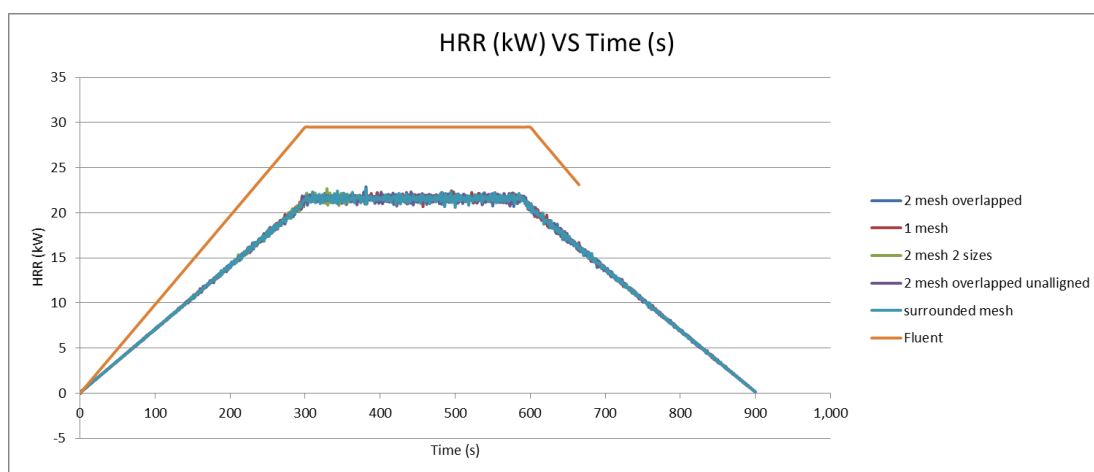


Figure 4.13: Comparison of HRR under Different Meshing Method with FLUENT

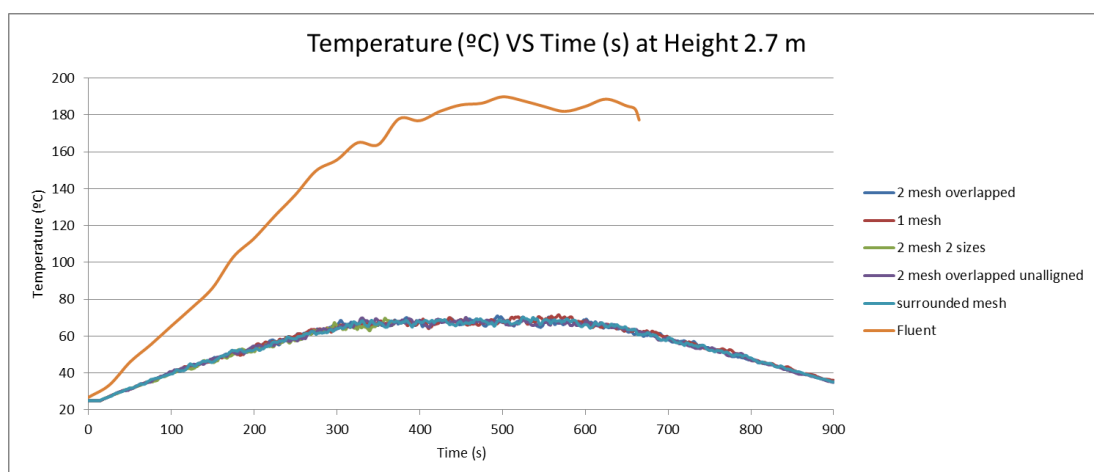


Figure 4.14: Comparison of Temperature under Different Meshing Method with FLUENT at Height 2.7 m

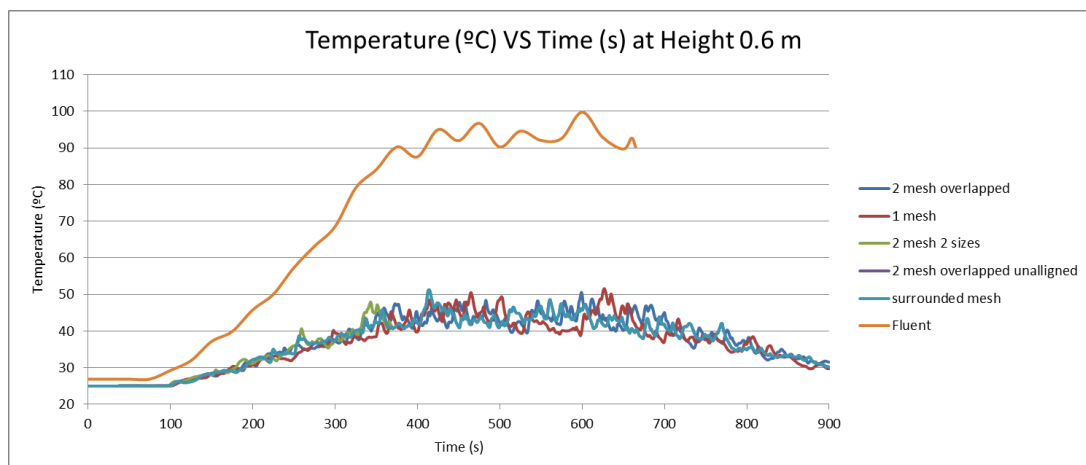


Figure 4.15: Comparison of Temperature under Different Meshing Method with FLUENT at Height 0.6 m

The HRR input is amended in FDS with convective heat flux turned on and rerun. Besides, the model in the FLUENT is also amended with the fire source area set to smaller, which is compatible with FDS model. The model in the FLUENT is also rerun. The simulation results are as below. From the previous meshing method selection in Section 4.2.2, since embedded mesh and 1 mesh are selected as appropriate meshing methods, therefore, only single mesh and embedded meshing methods are rerun with FLUENT for comparison.

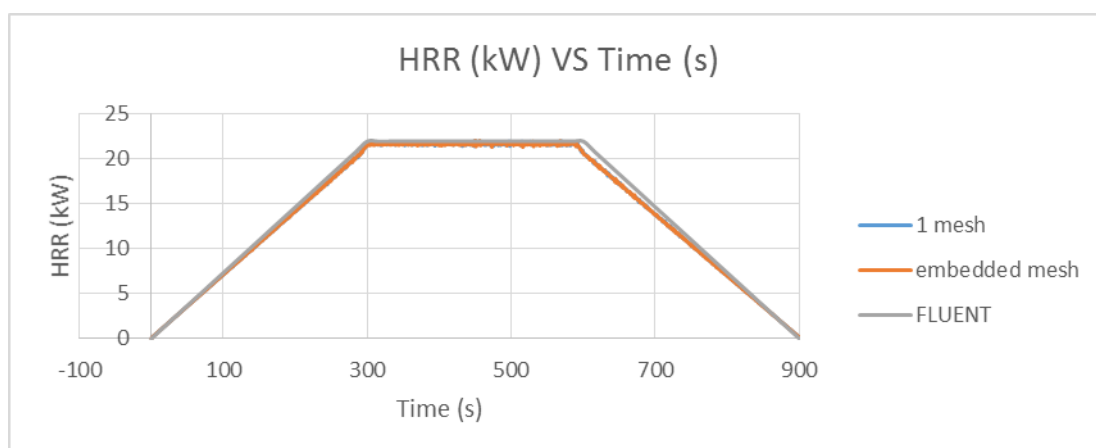


Figure 4.16: Comparison of HRR between FDS and FLUENT

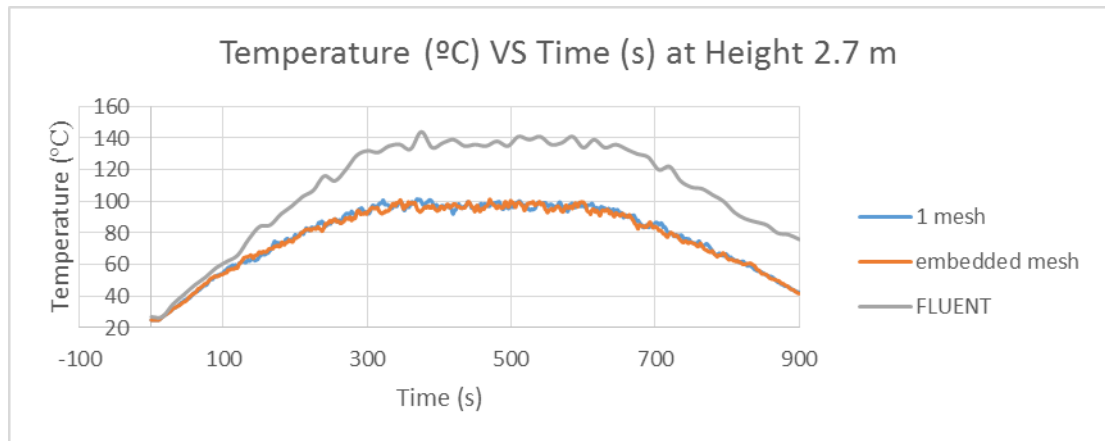


Figure 4.17: Comparison of Temperature between FDS and FLUENT at Height 2.7 m

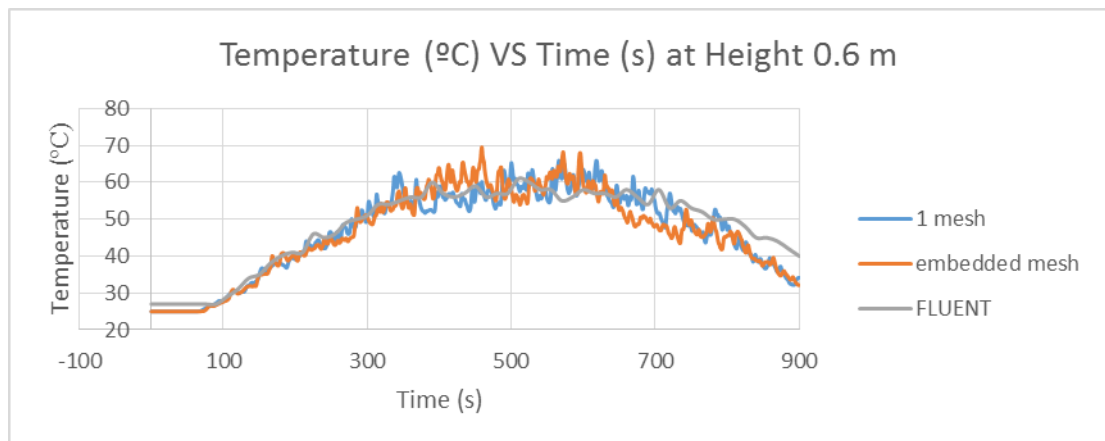


Figure 4.18: Comparison of Temperature between FDS and FLUENT at Height 0.6 m

From the Figure 4.16, the HRR curve for both FDS and FLUENT are close to each other after the adjustment of area of fire source inlet. Based on the point temperature near the curvature wall in Figure 4.17, at the height of 2.7 m of the model or the upper level, the temperature recorded by FLUENT is higher than FDS by about 40 °C. However, in Figure 4.18, the point temperature near the curvature wall recorded at height of 0.6 m for both FDS and FLUENT are very close to each other. With the reduced HRR in the FLUENT based on the previous model, the peak temperature had reduced as well. Furthermore, with the convective heat flux turned

on in the FDS, the peak temperature had also increased. In short, the variation of the FLUENT and FDS in temperature had decreased.

However, it is noted that the temperature predicted by both FDS and FLUENT at upper height level shows disagreement in prediction while at lower height level, the temperature prediction shows agreement in simulation results. To determine the hot and cold region of a room, graphs of height position against temperature are plotted.

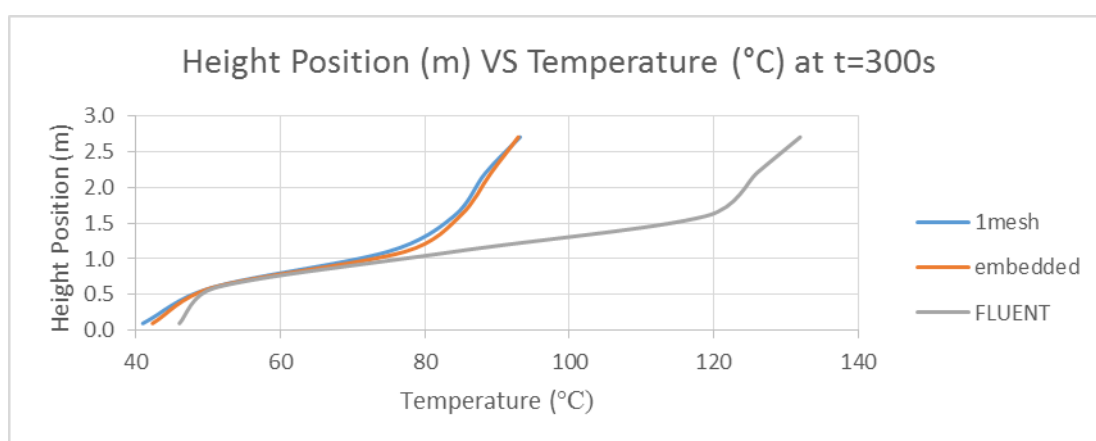


Figure 4.19: Hot and Cold Region Indication at Time = 300 s

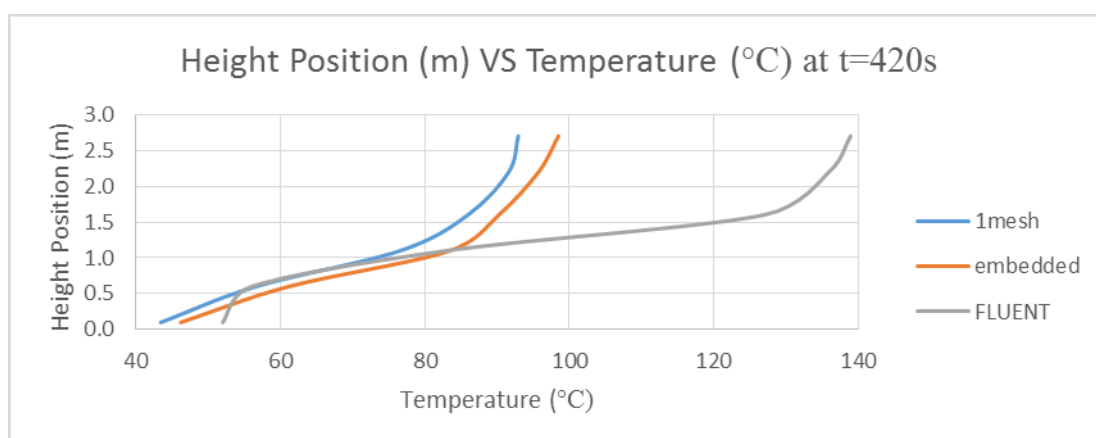


Figure 4.20: Hot and Cold Region Indication at Time = 420 s

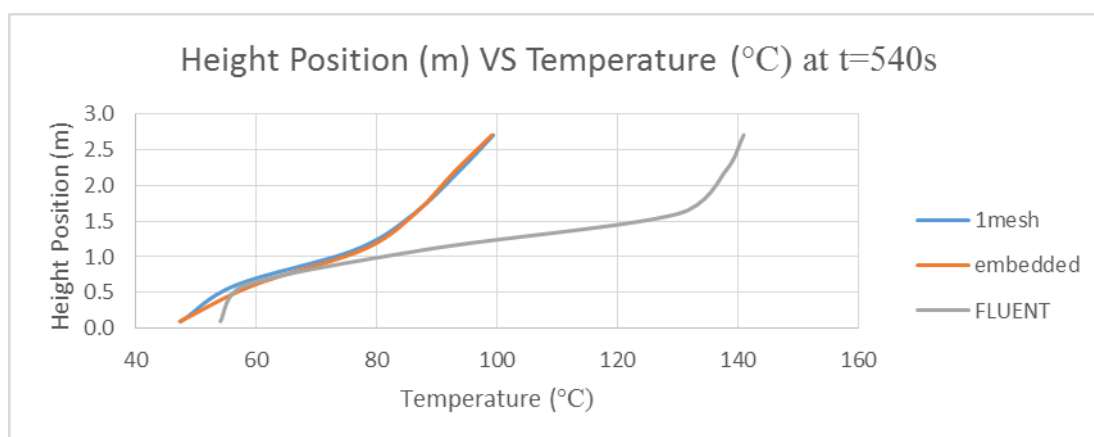


Figure 4.21: Hot and Cold Region Indication at Time = 540 s

The Figures 4.19 to 4.21 show that the hot and cold layers simulated by FDS and FLUENT at steady HRR state, which is at $t = 300$ s, 420 s and 540 s. At below 1.0 m of the model, the temperature prediction for both FDS and FLUENT are relatively close to each other. At below 1.0 m, this region is defined as cold region. Above 1.0 m, FDS and FLUENT simulation show variation where FLUENT has higher temperature prediction over FDS. At above 1.0 m, this region is defined as hot region. Comparing both FDS and FLUENT, both simulation shows agreement in the temperature prediction at cold region but shows disagreement in the temperature prediction at hot region.

One of the explanations that both FDS and FLUENT has different temperature prediction at hot region is that the hot air has not escaped out of the window in FLUENT. This is because in FLUENT, the windows in the room defined as pressure outlet, the hot air accumulates at the hot region in the room until the pressure builds up in the room and higher than the pressure gage of 0 Pa. At this moment, hot air flows out of the windows. Unlike in FDS, the windows are deleted off and the mesh is extended out of the structure. This creates an open boundary at the window where hot air can freely escape out of the windows. This explains the same results obtained by FDS and FLUENT at the height below 1.0 m of the room.

For the FLUENT simulation, the results may not be as accurate as FDS as the mesh cell size is coarser than FDS. In FLUENT, the mesh size is equivalent to the

FDS's fire characteristic diameter factor of 2. Furthermore, in FLUENT, the tetrahedron mesh is used, which provides less accuracy in simulation result. To improve the simulation result, the hexahedral mesh should be used and the mesh cell size should be sufficiently small enough, at least the size obtained with the fire characteristic diameter of factor 4. In addition, when simulation is done through FLUENT, the residual convergence is very poor, which the continuity residual drops up to $1e-03$ only.

Based on the temperature slice and velocity vector slice between FDS and FLUENT in Appendix F, the simulation results show similar result where the temperature layer in FDS and FLUENT is similar and only the magnitude is different. Similar result shown in the velocity vector where the direction of flow field is similar but with different magnitude. To examine the two-way flow at the window in FDS, this method is not able to be used as the result cannot be displayed in velocity vector at the window pane. However, the FDS has air flow monitoring device to measure the air flow.

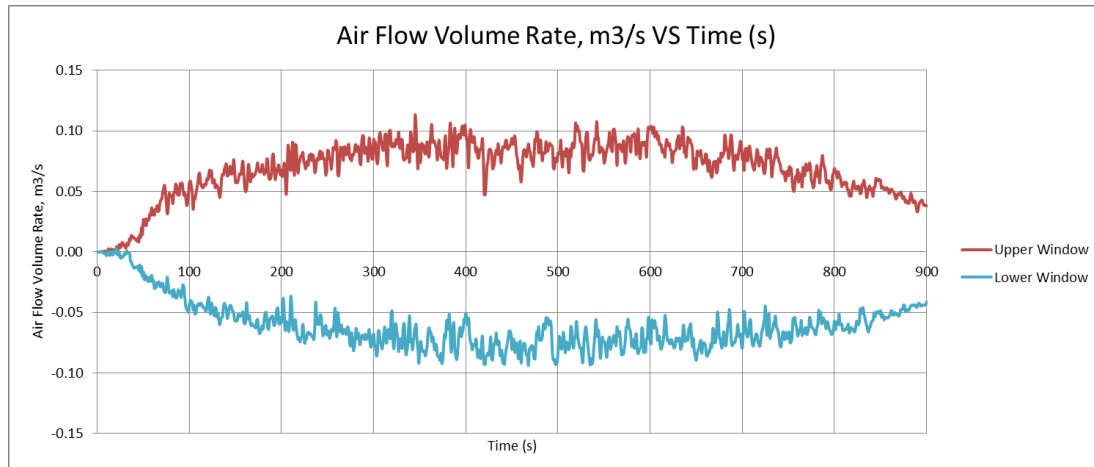


Figure 4.22: Air Flow at Window by FDS

At the window, two air flow monitoring device are placed, one at the upper region while another at the lower region. From the Figure 4.22 above, it is shown that there is two-way flow at the window in FDS fire simulation. The positive air flow volume rate shows the air flowing out of the room while negative air flow is vice versa. However, the air flow monitoring device only measure the air flow of a

specified area, which the inflow and outflow area must be ascertained so that the air flow monitoring device can measure the air flow accurately. For this reason, the two-way flow is not used to compare the simulation result between FLUENT and FDS.

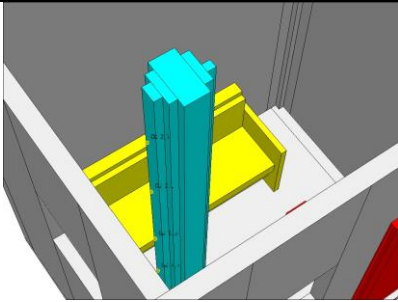
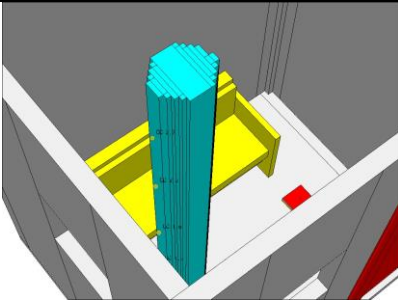
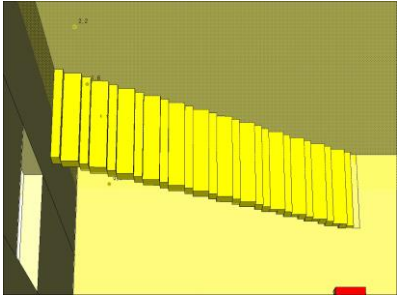
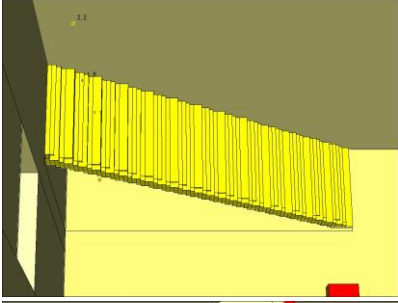
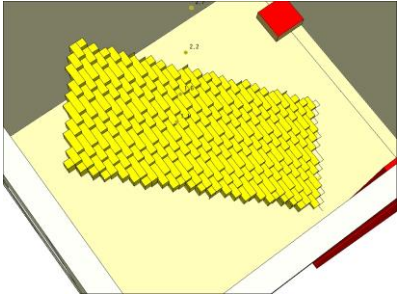
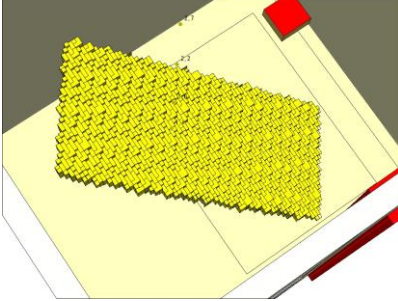
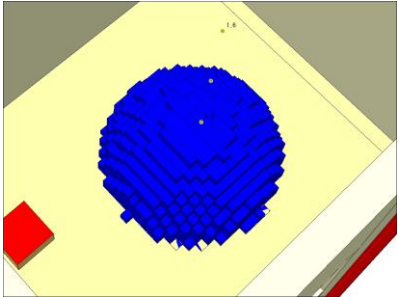
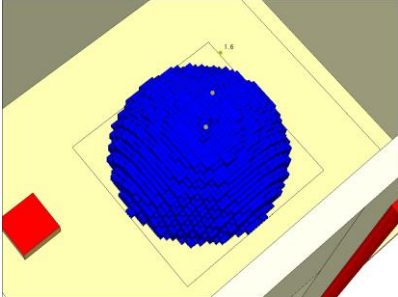
Recall back in the Chapter 2 Methodology, the research done by Wang BinBin on the Comparative Temperature between FDS and FLUENT, FLUENT had over predicted the temperature when the simulation is verified and validated with FDS and experimental data. From this research, the simulation done by both FDS and FLUENT have similar trend to Wang BinBin, which is in agreement with the simulation result with the past research.

4.2.4 FDS Simulation with Various Curved Geometries

As discussed in the Methodology, a few curved geometry have been simulated as shown in Table 4.4. These simulations on the curved geometry is done to study how the embedded mesh can be applied in every complex geometry. Furthermore, the simulation results are compared with the base model where the Method 1 (1 mesh) is applied to ensure there is no significant difference in the simulation results.

Based on various curved geometry such as curved wall, cylindrical column, ramp and hemisphere, the simulation results shows that with the application of embedded mesh on the curved geometry, the stepped curved surface can be smoothen. This can be done without affecting the simulation results like temperature and velocity flow field. The simulation results are obtained at 400 s and 500 s of the fire because at these specific time, the HRR of the fire is at steady state. Based on the velocity vector, temperature slice and temperature isosurface in Appendix G, the simulation results show no significant difference between 1 mesh method and embedded mesh method.

Table 4.4: Comparison of Curved Geometry between 1 Mesh and Embedded

Curved Geometry	Mesh	
	1 Mesh	Embedded Mesh
Cylindrical Column		
Ramp		
Ramp Unaligned		
Hemisphere		

For the point temperature comparison, the temperature is compared at the height level of 0.6 m and 2.7 m of the structure for each curvature geometry. A set of thermocouple tree is fixed near to the curvature geometry to record the temperature. From the Figure 4.23 to Figure 4.26, the temperature shows consistency for both 1 mesh and embedded mesh method. The temperatures are close to each other for both

method. Therefore, the use of embedded mesh will not cause a large variation in the simulation results.

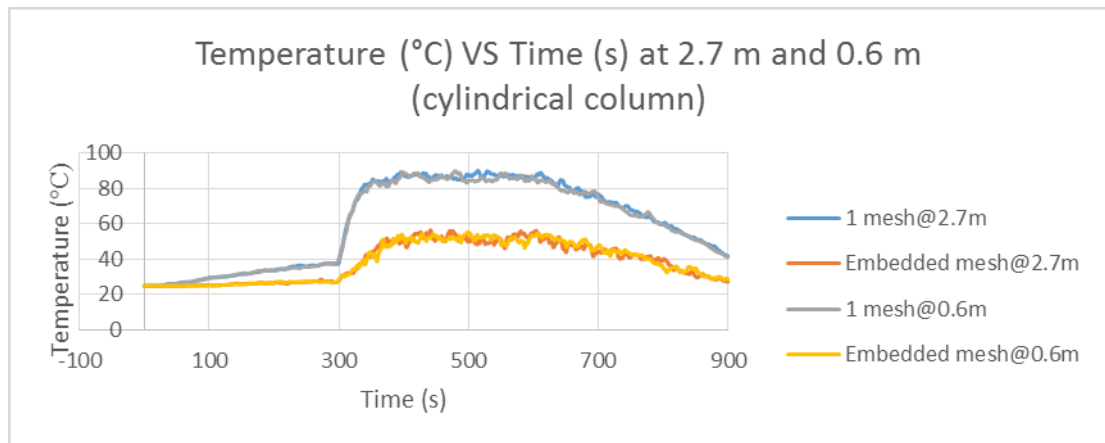


Figure 4.23: Comparison of Point Temperature between Meshing Method for Cylindrical Column

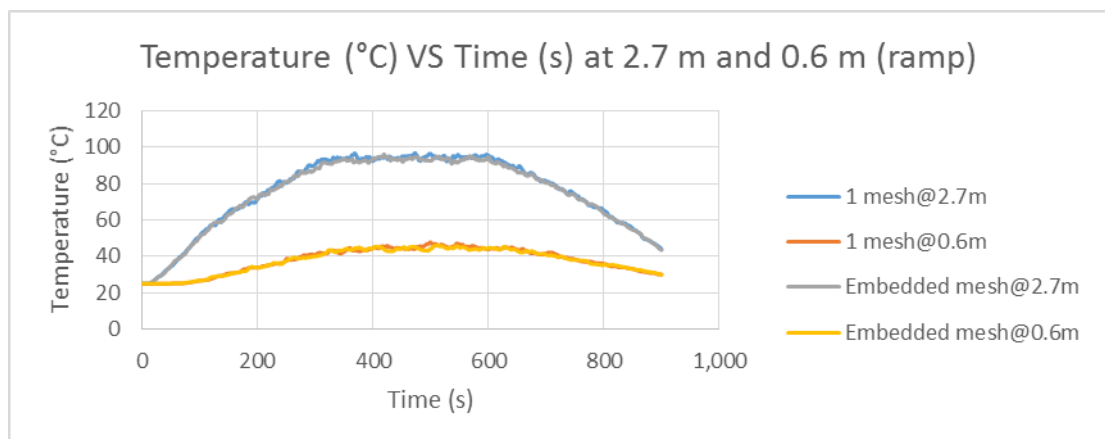


Figure 4.24: Comparison of Point Temperature between Meshing Method for Ramp

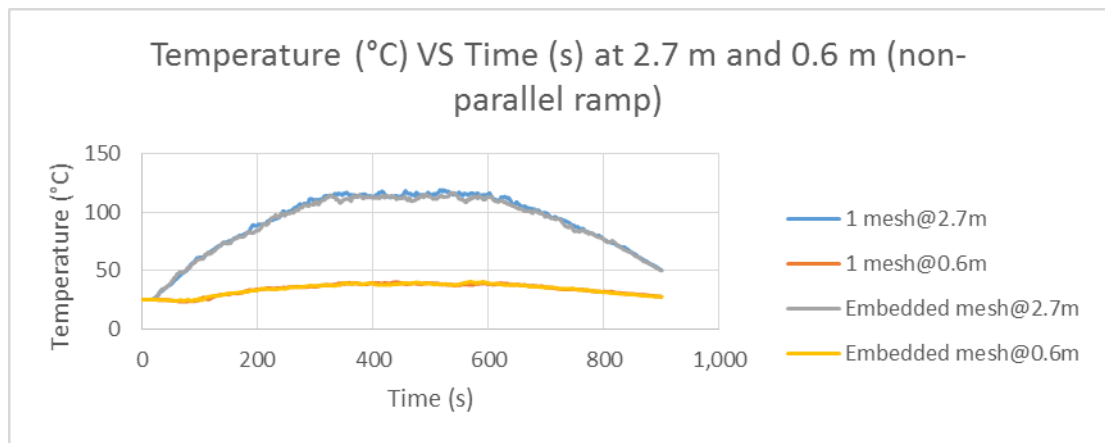


Figure 4.25: Comparison of Point Temperature between Meshing Method for Ramp (non-parallel)

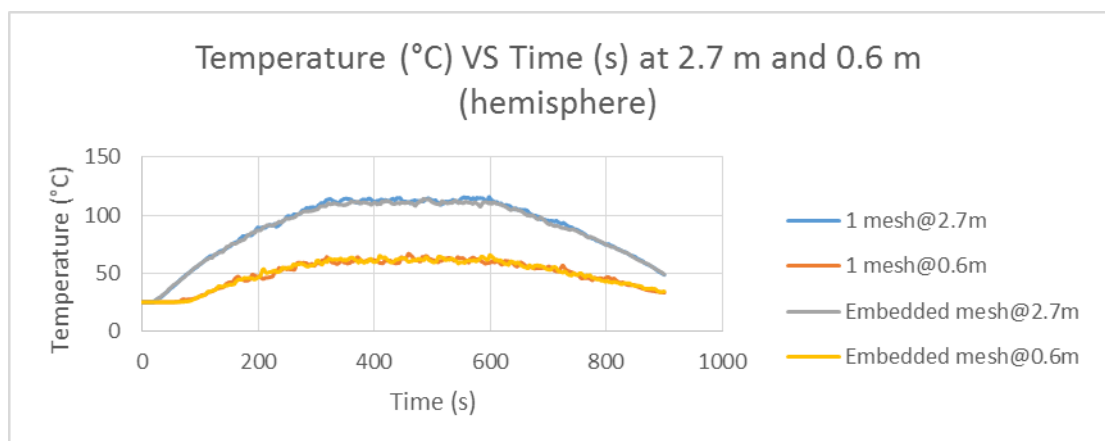


Figure 4.26: Comparison of Point Temperature between Meshing Method for Hemisphere Structure

Although Method 3 (Embedded mesh) is proven as the appropriate meshing method on generating mesh on curved geometry, there is a limitation of using this method. In solving the embedded mesh in FDS simulation the inner mesh actually retrieves information from the outer mesh which is also known as primary mesh. The primary mesh receives no information from other mesh. In other words, the calculation in the embedded mesh is dependent on the primary mesh while the calculation in primary mesh is independent. In calculation, the embedded mesh receives information from primary mesh but the calculation in embedded mesh is not

sent to the primary mesh. In other words, whatever captured in the embedded mesh is not reflected to the primary mesh.

The initial purpose of the embedded mesh is to enhance the resolution of a zone of interest of a building, for example, to study the fire spread in a room of a house. However, in this research, the embedded mesh is implemented just to smoothen the curvature surface where the stepped curved geometry is reduced. It shows no problem on the numerical stability issue. However, this method requires extra precaution and the simulation results can be questionable. From the FDS online discussion group, the experts also advise the use of single fine mesh, provided if the large computational resources available.

For this reason, Method 1 (1 mesh) is still a preferably method in meshing the geometry although the curvature geometry representation is compromised. As discussed in the methodology earlier on the reduction factor study of embedded mesh, this study is not done in this research due to the numerical solving issue faced when solving embedded mesh.

4.2.5 FDS Simulation with Various Type of Fire

From the three types of fire, only typical fire and wall fire are able to be run. Corner fire encountered numerical instability and this is most probably due to the high velocity in the flow field which cannot satisfy the CFL condition. Based on the plot3D in Figure 4.27, the results show high velocity at the fire inlet and at the windows, which is recorded at 225 ms^{-1} . with this high velocity, the simulation stopped due to numerical instability.

There are few methods to examine the characteristic of fire based on each type of fire, which are the measure of incident heat flux, horizontal flame spread and vertical flame spread. In this research, the vertical flame spread is used and the vertical velocity of the fire are compared among each type of fire. (Cheng 1995)

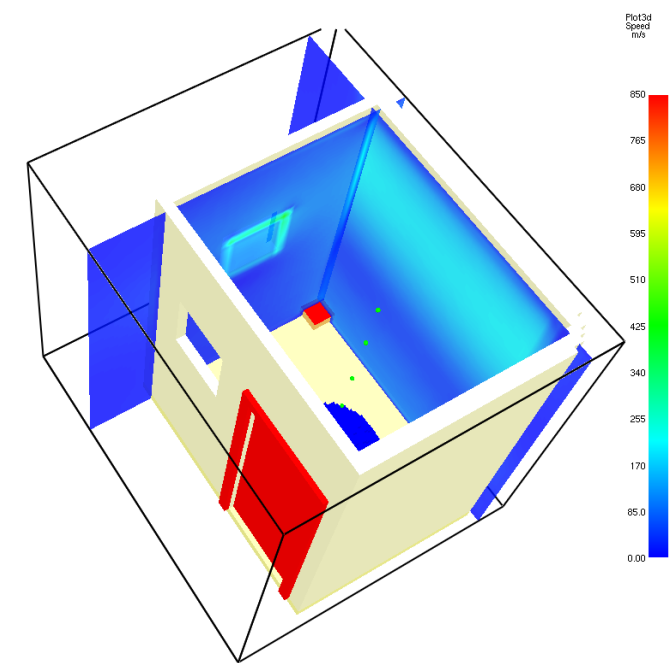


Figure 4.27: Plot3D Velocity View of Corner Fire

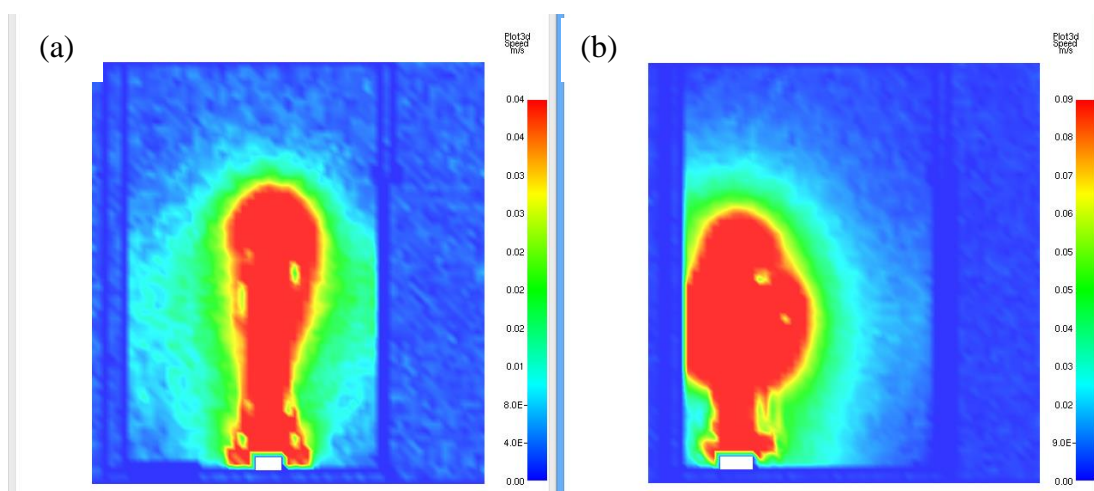


Figure 4.28: Velocity Comparison between Typical Fire (a) and Wall Fire (b) at Time = 4 s (plane slice)

From the Figure 4.28 above, at the beginning stage of the fire (4 s), the typical fire (left) has lower flame speed compared with wall fire (right). Based on the value of the flame speed, the wall fire has twice faster flame speed than typical fire. This is because in HRR, the heat is release to the surrounding in hemispherical region. In wall fire, the hemispherical region has been reduced by half, thus resulting

in higher flame speed. At the steady state of the fire (400 s), it is also shown that the flame speed of wall fire is more vigorous than the typical fire, which is shown in Figure 4.29 below.

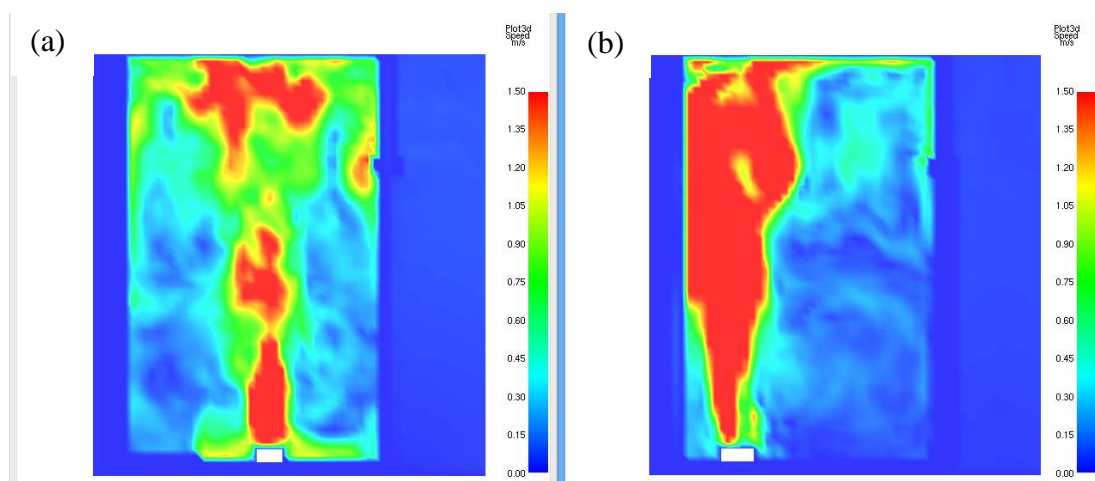


Figure 4.29: Velocity Comparison between Typical Fire (a) and Wall Fire (b) at Time = 400 s (plane slice)

From this simulation, it is proven that the FDS is able to simulate different fire scenarios, which are the typical fire and wall fire. The simulation results show agreement with the expectation, where the fire spread faster in wall fire as determined by the fire speed. This simulation is helpful because in fire reconstruction, there may be many fire scenarios can be simulated and FDS proved the capability of simulating the typical fire and wall fire in a realistic situation.

4.3 Fire Model DNS Simulation Using PyroSim

For the DNS simulation type, the simulation results show significant different from the LES simulation type. From the SmokeView shown in Figure 4.30, the incident heat flux shows the different result compared with LES model where the incident heat fluxes recorded at the walls are lower compared with LES model. On the animation of the particle tracer, the simulation results shows similar pattern

compared with LES model but it is noticed that the tracer does not travel as vigorously as the LES model.

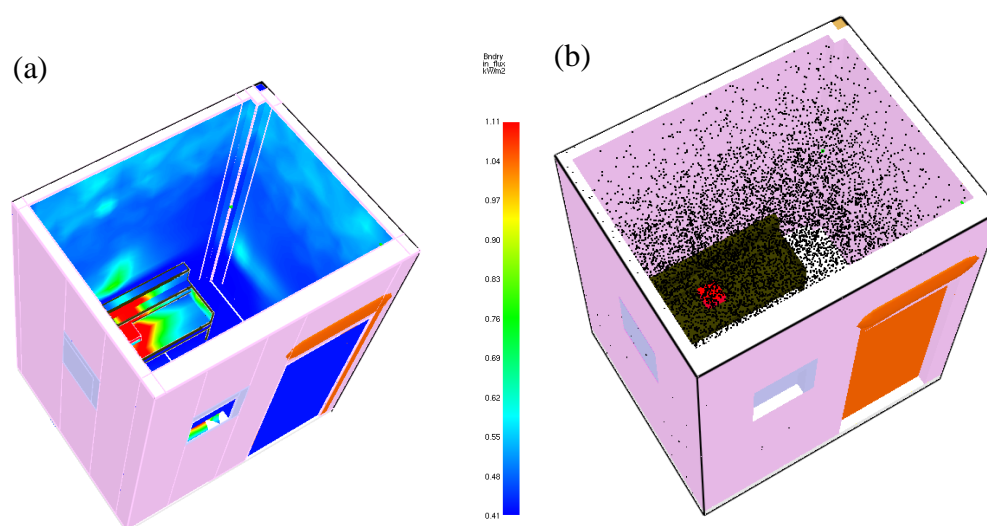


Figure 4.30: DNS Simulation Result of (a) Incident Heat Flux and (b) Fire Tracer

From the Figure 4.31, the LES and DNS models shows difference in the temperature profile at time 100 s. The two models were simulated under same input parameters where the meshing size is the same. It is noted that the DNS models shows a hot region at the window and the temperature profile does not show full temperature slice. The incomplete temperature slice is most probably due to the insufficient computational power during simulation to process and store the simulation data. The fire source input was rechecked but no error was found. The red region at the right side is not defined as fire source inlet but the simulation results shows a sign of fire originated from the windows.

On the other hand, the temperature profile shows at the fire source inlet, the heat release is smaller compared with LES model. The variation is most probably due to the direct switching of LES to DNS simulation type. In order to do simulation using DNS, the mesh size should be refined small enough to capture the small eddies of turbulence. However, in the methodology, the mesh size is not refined when trying to solve the simulation using DNS and thus causing the inability of the simulation to

be done more accurately under DNS mode. When the mesh size is refined to smaller size, more cells are actually created and thus increase the computational power required to solve the simulation which is unlikely to be done in the conventional computer as explained in the PyroSim User Manual. To use DNS, higher computational computer like workstation or mainframe should be used.

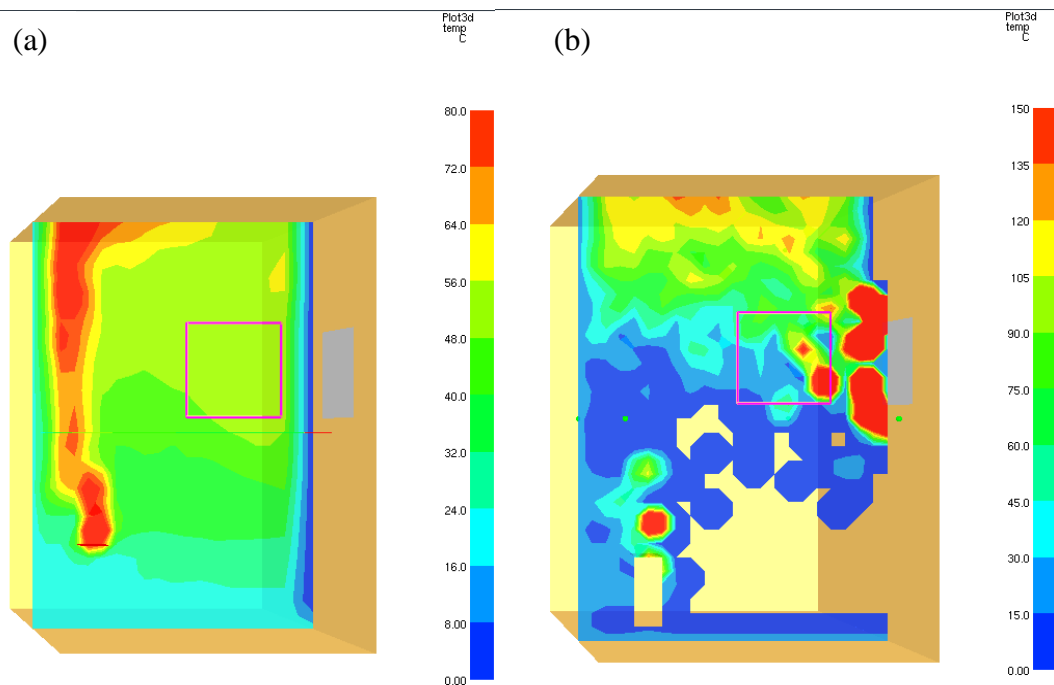


Figure 4.31: Comparison of Temperature Profile between (a) LES and (b) DNS Simulation Model

To determine the number of mesh cells required to run the DNS so that the turbulent eddies can be captured, the equation is as below.

$$N \sim \frac{L}{\eta} \quad (4.1)$$

where

N = number of grid points required

L = length scale of the largest turbulent eddy

η = length scale of viscous effect

In fire model simulation, DNS simulation is a better than LES simulation in testing the basic solver, especially when the solution is at steady state, provided there is a high computational power computer. Comparing the HRR curve between LES and DNS model, it is found that the DNS's HRR does not fluctuate much. It can be concluded that the fire model simulated in DNS is more towards steady state which can be shown in the first 40 s in Figure 4.32 below where the HRR does not fluctuate much.

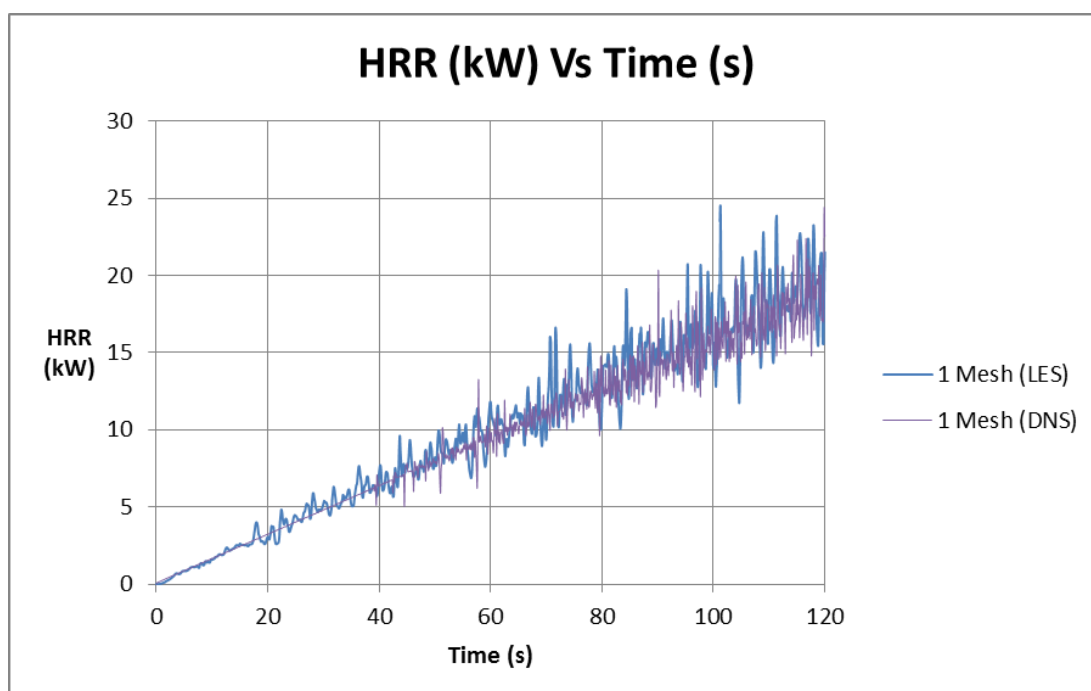




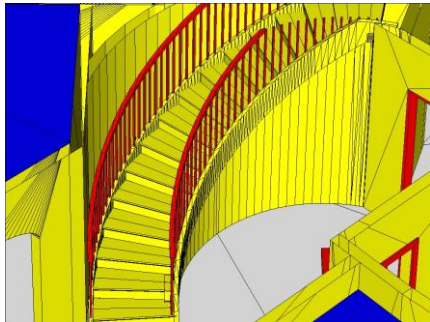
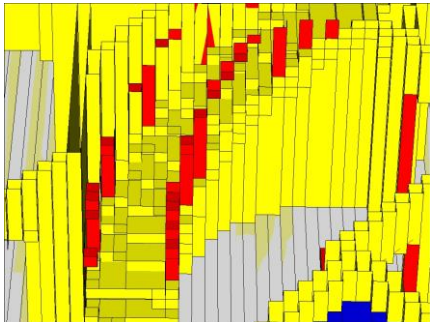
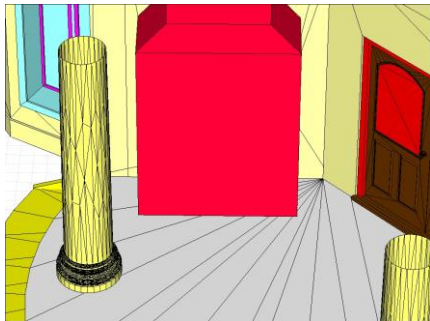
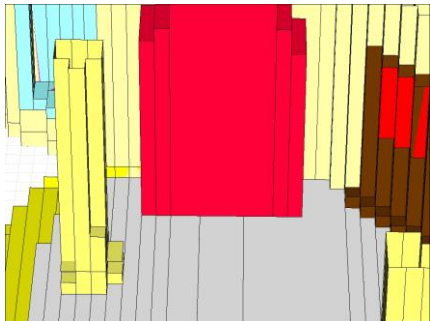
Figure 4.32: Comparison of HRR for LES and DNS

The difference in DNS and LES modelling is mainly due to the high computational resource required in DNS simulation. Both DNS and LES model are done in an ordinary midrange laptop and thus the computational power to model DNS may be insufficient. As described in Fire Dynamics Simulator 5 Technical Reference Guide, the DNS simulation is not possible to be run in the conventional computer. Therefore, the DNS simulation type is not readily available to be done in this research using the ordinary midrange laptop.

4.4 Full Scale Model Simulation by FDS and Walkthrough in SmokeView

When the full scale house is imported into the PyroSim, under meshing size of 0.15 m rectilinear cells, the model is represented in FDS blocks. However, some of the geometry is not well represented in FDS blocks, which is shown in the Table 4.5 below due to the coarse mesh used. The FDS blocks are created with rectilinear mesh in such a way that when an object occupied more than half of the volume of the mesh cube, obstruction is created in that particular cube, forming FDS blocks.

Table 4.5: Comparison of Actual CAD Model and FDS Block

	Actual CAD Model	FDS Block
Roof		
Staircase and Railing		
Cylindrical Column		

The simulation is done in three days without numerical instability issue. The smoke propagation can be displayed for the entire house, which can be view through

SmokeView. From the simulation result in PyroSim, the simulation output is not able to be exported back to Revit Architecture at this moment. The simulation outputs produced by the PyroSim are in the format of .q, .s3d, .sz, .sf, .prt5 and other .csv files, which is not able to be read by Revit Architecture. The simulation results in FDS are actually stored in the nodal point of the mesh. To enable to view the simulation results, the data is extracted from the nodal point of the mesh and displayed in SmokeView. To view the simulation result in Revit Architecture, the simulation output from FDS should be extracted and transferred back into Revit Architecture.

In the new FDS 6, there is one FDS extension, `fds2ascii.exe`, which is able to extract data from simulation result. The `fds2ascii.exe` is able to extract parameters like temperature, velocity, heat flux, etc from three main sources of data, which are the Plot3D file (PL3D), Boundary file (BNDF) and Slice file (SLCF). The `fds2ascii` is able to extract these data for a period of simulation time and then export into a formatted text, which can be opened by Microsoft Excel. In other words, the `fds2ascii.exe` converts the unformatted binary file from FDS into ASCII csv file. This attempt of exporting the simulation back to Revit Architecture failed as the data extracted is not able to be read by Revit Architecture. The `fds2ascii.exe` functions to extract the data from slice file into formatted text file which can be analysed by other software.

The effort to export the simulation result is still lying on how the simulation output files are created. Based on the FDS source code, which is available in the FDS online discussion group, “`dump.f90`” is the simulation output file generation, which generates the output data file as the simulation goes by. The source code is created based on the FORTRAN code, which can be edited in the FORTRAN compiler. For the SmokeView to display the 3D smoke and fire, semi-transparent planes of fire and smoke are drawn in the SmokeView. The smoke and fire data are created by FDS at fixed time intervals using endian flag, which is a common method of storing the data in memory. Based on the sectional source code in “`dump.f90`” as shown in Figure 4.33, the source code shows how the smoke and fire data files are created. (NIST 2012)


```

! header

WRITE (LU_SMOKE3D) ONE, VERSION, 0, NX-1, 0, NY-1, 0, NZ-1

! time depended data

WRITE (LU_SMOKE3D_SIZE, *) TIME, NCHARS_IN, NCHARS_OUT
WRITE (LU_SMOKE3D) TIME
WRITE (LU_SMOKE3D) NCHARS_IN, NCHARS_OUT
IF (NCHARS_OUT > 0) WRITE (LU_SMOKE3D) (BUFFER_OUT (I), I=1, NCHARS_OUT)

```

Figure 4.33: Source Code of Smoke and Fire Output (NIST 2012)

For suggestion, the export of simulation file into Revit Architecture can be done, provided that the simulation output file format can be read by Revit Architecture. This involves programming the code to alter the output file format, which requires high programming skills and knowledge. At this moment, it is not possible to export the simulation output file into Revit Architecture without reprogram the source code.

However, in PyroSim, the walkthrough is successfully to be created in the SmokeView. This is done by entering the coordinates of the keypoints which will then resembles to a line or path of the walkthrough. While in the walkthrough mode, the simulation result of fire and smoke are also able to be run together. The real-life person walkthrough in the fire event is able to be created, which the viewing of growth of fire and smoke layer is able to be done.

The walkthrough in the PyroSim can be viewed in the SmokeView but there is no saving option for the walkthrough. Unlike Revit Architecture, the walkthrough can be saved as video file which can be opened by Windows Media Player. For PyroSim, the walkthrough created is saved together with the simulation result. Based on the walkthrough created, there are two ways of displaying the geometry file, either in actual CAD model or in FDS block which is shown in Figure 4.34 and 4.35. In both displaying methods, the smoke can be displayed but the soot data is obtained based on the mesh cells. This results in staggered soot display even the geometry model is displayed in actual CAD model.

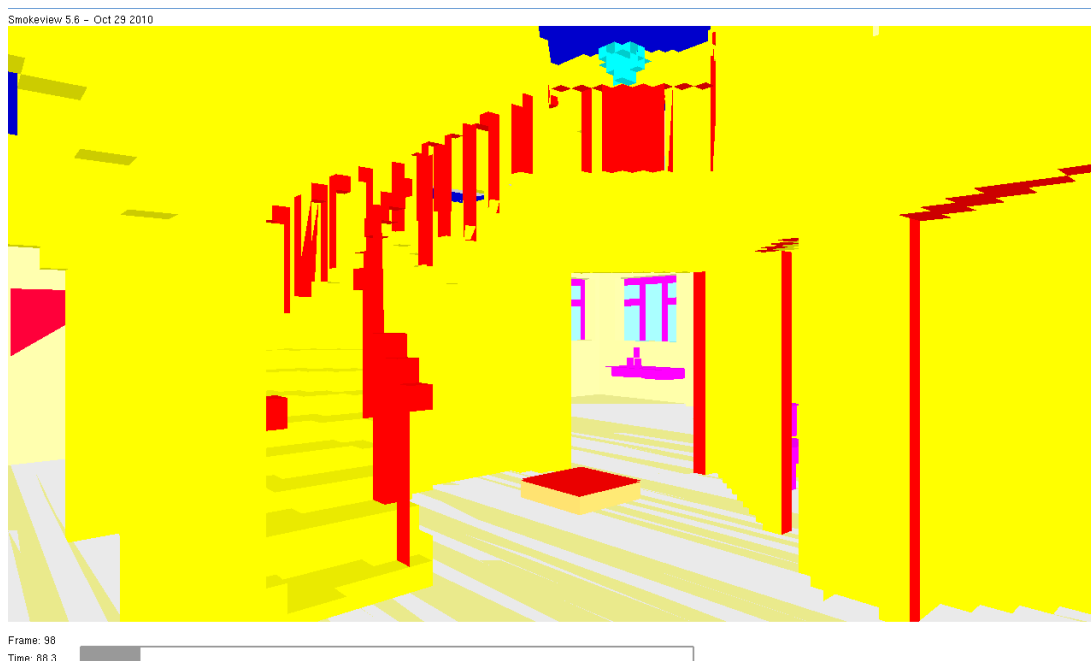


Figure 4.34: Interior FDS Block Display in SmokeView

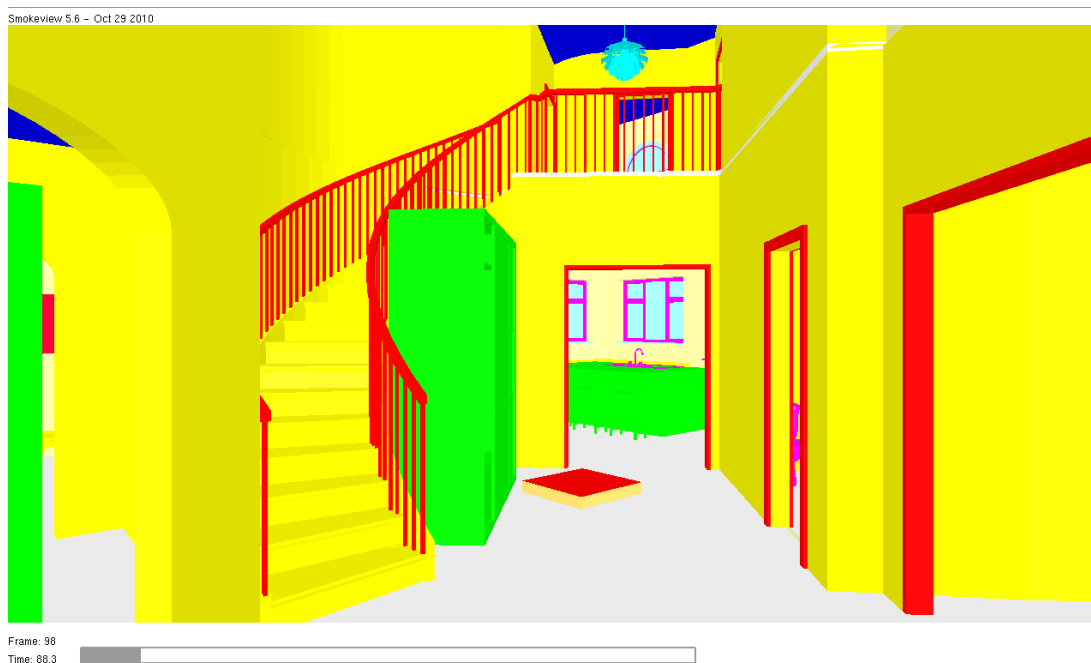


Figure 4.35: Interior Actual CAD Model Display in SmokeView

For the walkthrough in SmokeView, the soot and fire are turned on while the walkthrough is being played. To record the walkthrough, another software, FRAPS is installed and run parallel with the SmokeView. The FRAPS is used because the

SmokeView cannot save the walkthrough as video file. The FRAPS captures the screen and converts into video. With SmokeView and FRAPS running simultaneously, the walkthrough captured does not have smooth frame rate but rather lagging. To enhance the walkthrough captured, the video file is edited in the Windows Movie Maker and the video file is switched to fast motion speed so that the walkthrough looks smooth with the frame rate.

CHAPTER 5

CONCLUSION AND RECOMMENDATIONS

5.1 Research Summary

At the end of the Project, a simple room is drawn in the Revit Architecture for simulation. The drawing is then exported to FLUENT as “.sat” and to PyroSim as “.dxf”. The structural geometry is successfully linked between Revit Architecture to FLUENT and PyroSim. A walkthrough is created in Revit Architecture. The linkage from Revit Architecture to PyroSim and FLUENT is created.

For the fire modelling simulation in FLUENT, the four models which are using heat flux as fire source and using non-premixed combustion are done. For the first heat flux model, the fire simulation does not show diffusion type fire but rather a heat transfer to the surrounding as the temperature rise is low. For the second heat flux model, the Boussinesq model, which solves the thermal buoyancy turbulent flow is used. The simulation output shows a diffusion type fire and the temperature rise is also reasonable. Using UDF, the HRR ramp-up, steady state and decay period can be simulated throughout the fire scene.

For the third non-premixed combustion model, the fire with chemical reaction is successfully created. However, the major problem faced in the FLUENT is the boundary condition which is the auto-define on the boundary condition. The fire from the fire inlet is more likely a jet stream of fire which is not a diffusion rate type of fire; this jet stream of fire simulated is not the desired fire type. The fire simulated using non-premixed combustion is more suitable for furnace, boiler, fire jet, which is

not suitable to be used for normal fire event. Despite the fourth non-premixed combustion model using UDF to create a pyramid velocity profile for the fire inlet, the fire simulated is still like a jet stream of fire. Furthermore, there are too many variables to be set for non-premixed combustion model such as fuel type, air fuel mixture ratio, velocity inlet, etc. These factors are the difficulties when simulating the fire and incompatible to be compared with FDS's result.

For the fire modelling simulation in FDS, five methods of meshing the geometry are tested and simulated using LES simulation type. To ensure the accuracy of the simulation result, the fire diameter characteristic equation is used as a guidance to determine the mesh cell size to be used. In this research, the factor of characteristic fire diameter used is 4, which is sufficiently good enough to simulate good accuracy in fire simulation. The main limit of using higher factor is the limitation on the computational resources available. The precaution steps must be taken to avoid the numerical instability issues faced in the FDS so that the simulation can be done successfully. All five meshing methods are run and the results are compared among each other in terms of point temperature, temperature slice, temperature isosurface, velocity vector slice and HRR check. In addition, the computational storage, computational duration, ease of meshing, prone to numerical instability and curved surface meshing quality are also compared.

All five methods show that the simulation results are close to each other in regardless of meshing method. However, when more than one mesh is used, the simulation is more prone to numerical instability. For the embedded mesh method, the simulation is less likely to encounter numerical instability but there is solver issue for the embedded meshing method, which may lead to questionable results. For this reason, a single fine mesh is found to be still preferably and suitable over the other methods.

When the simulation results are compared between FDS and FLUENT for data verification, the heat flux with Boussinesq model is used as this model is the most compatible model simulated among four models in FLUENT. Based on the point temperature comparison, FLUENT over-predicted the temperature at height above 1.0 m than FDS but both FDS and FLUENT shows similar temperature curve

at below 1.0 m of the room. FLUENT's simulation results may be inaccurate as the mesh size is coarse and the residual is bad, which may lead to the over prediction in temperature. The discrepancy in temperature at hot layer may be also due to the boundary condition specific as pressure outlet at the windows. However, based on the velocity vector slice and temperature isosurface, Both FDS and FLUENT shows similar results but different magnitude, which means there is no significant difference in the flow field in the fire simulation done in both FDS and FLUENT. This also proves that FLUENT is capable of simulating fire.

For the DNS in FDS, this option is still not readily available as the computational power required to run DNS is very high. This is because the mesh must be refined to small enough to trace the turbulent eddies, which generates large number of mesh cells. Based on the workstation used in university, the workstation is unable to handle DNS simulation. For this reason, the LES is used instead of DNS.

For the large scale model simulation done for a whole unit of house, the simulation is done on one single fine mesh with mesh size flowing the characteristic fire diameter factor of 4. From the simulation, a walkthrough is done together with fire and smoke to illustrate the real life fire scenario walkthrough. The walkthrough is done in SmokeView where the keypoints are entered to form a path. To enhance the geometry display in SmokeView, the CAD model display can be turned on instead of viewing coarse FDS block.

The attempt of exporting the simulation results back into Revit Architecture is also tired. By default, Revit Architecture is unable to read any of the simulation output files. One of the attempt is by using `fds2ascii.exe` which extract the data of a slice file and saves as formatted text file. However, Revit Architecture is unable to read such output file from FDS. To export the simulation output file into Revit Architecture, the data at each nodal point should be extracted and transferred to Revit Architecture. At the same time, the data format of the simulation output must be also readable by Revit Architecture.

The effort of exporting the simulation result is still depending on how the simulation output files are created. Based on the FDS source code, the "dump.f90" is

the source code which involves in creating the simulation output files. To ensure the simulation output file is readable by Revit Architecture, the source code “dump.f90” should be edited to change the output file format. However, the alteration of source code is complicated and requires high level of programming skill. In this research, the export of simulation output back in to Revit Architecture is not successful.

In short, the aims and objectives stated in Chapter 1 are achieved except the exportation of the simulation result into Revit Architecture. The effort of exporting simulation result into Revit Architecture requires further study.

5.2 Recommendations for Future Research

For the comparison of simulation results between FDS and FLUENT, the data verification is still questionable as the simulation results are not able to be confirmed whether output is correct. To validate the simulation results, real fire experiment should be carried out. With fire experiment, the simulation results can verify with actual experiment data.

To improve the computational power of the workstation, several workstations should be clustered together. With clustered processing workstation, the computational power can be greatly improved. Using FDS, the MPI simulation can be turned on to fasten the simulation. This MPI feature is available in FDS other than OpenMP feature. With high computational resources available, using one single mesh, the mesh cell size used can be very fine, where the characteristic fire diameter factor can go beyond 10. This results in higher accuracy in simulation as well as high quality meshing on curved geometry, which the stepped surface due to rectilinear block is reduced on curvature surface.

For the export of simulation output into Revit Architecture, the source code needs to be further study to understand how the programming code is involved in the simulation output file which can be read by Revit Architecture. This requires high

understanding and programming skill in order to edit the source code in the FORTRAN compiler.

Besides, the features in the FDS should also be explored more such as the smoke detector, heat detector, chemical-reacting fire, burn away option by mass loss, sprinkler nozzle and EVAC. With these features, a more realistic fire scenario, which involves evacuation of victims can be created. This will allow the study of how the geometry of the structure can affect the fire spread, smoke propagation as well as the evacuation plan of the victims. This also allows architects, fire safety engineers, mechanical and structural engineers to corporate together to improve fire safety design in building projects.

REFERENCES

- Andersson J. and Lennqvist A., 2010. *A burn-through model for textile membranes in buildings as a tool in performance based fire safety engineering*. Published final year project. Lund University, Sweden.
- ANSYS Inc. 2009. *ANSYS FLUENT 12.0 Theory Guide*. [online] Available at: [https:// www.sharcnet.ca/ Software/Fluent12/pdf/th/flth.pdf](https://www.sharcnet.ca/Software/Fluent12/pdf/th/flth.pdf). [accessed 13 June 2012]
- ANSYS INC. 2010. *ANSYS 13.0 Meshing User's Guide*. [online] Available at: [http://www1.ansys.com/ customer/content/documentation/130/wb_msh.pdf](http://www1.ansys.com/customer/content/documentation/130/wb_msh.pdf). [accessed 26 July 2012]
- Banerjee D. et al., 2009. *Visualization of Structural Behavior under Fire*. National Institute of Standards and Technology.
- Cha M. et al., 2012. A Virtual Reality Based Fire Training Simulator Integrated with Fire Dynamics Data. *Fire Safety Journal* 50, 12-24. [online] Available at: <http://www.sciencedirect.com/science/article/pii/S0379711212000136>. [accessed 15 January 2013]
- Chai N. and Chow W.K., 2012. Numerical Studies on Heat Release Rate in Room Fire on Liquid Fuel under Different Ventilation Factors. The Hong Kong Polytechnic University.
- Cheng Q., 1995. *Turbulent Flame Spread on Vertical Corner Walls*. University of Kentucky.
- Choi J., 2008. *Concurrent Fire Dynamics Model and Thermomechanical Analysis of Steel and Concrete Structures*. Published final year project. Georgia Institute of Technology.
- Chow W.K. and Zou G.W., 2004. Correlation Equations on Fire-induced Air Flow Rates through Doorway Derived by Large Eddy Simulation. *Building and Environment* 40, 897-906. [online] Available at: [http://www.sciencedirect.com/science/ article/pii/S0360132304002781](http://www.sciencedirect.com/science/article/pii/S0360132304002781). [accessed 13 June 2012]
- Domingos X., Viegas and Luis P.P., 2004. *Fire Spread in Canyons*. Published final year project. Department of Mechanical Engineering, University of Coimbra, Portugal.

- Fire and Rescue Department Malaysia (JBPM), PAM, IEM, ACEM. 2006. *Guide to Fire Protection in Malaysia*. United Kingdom: The Institute of Fire Engineers (UK) Malaysia Branch (IFEM).
- Gutierrez-Montes C. et al., 2008. Experimental Data and Numerical Modelling of 1.3 and 2.3 MW Fires in a 20m Cubic Atrium. *Building and Environment* 44, 1827-1839. [online] Available at: <http://www.era.lib.ed.ac.uk/handle/1842/2761>. [accessed 3 August 2012]
- Heinisuo M., Laasonen M. and Outinen J., Fire Design in Europe. [online] Available at: http://people.fsv.cvut.cz/www/wald/COST_C26_Prague/pdf/51_Fire%20desig n20in%20Europe_sm.pdf. [accessed 13 June 2012]
- Hong Y. et al. *Geometry-based Control of Fire Simulation*. Unpublished journal. Institute of Computing Technology Chinese Academy of Science, Beijing, China.
- Hu L.H., Huo R. and Yang D., 2008. Large Eddy Simulation of Fire-induced Buoyancy Driven Plume Dispersion in an Urban Street Canyon under Perpendicular Wind Flow. *Journal of Hazardous* 166, 394-406. [online] Available at: <http://www.sciencedirect.com/science/article/pii/S0304389408017238>. [accessed 3 August 2012]
- Kevin B. et al., 2010. Thermal Radiation from Large Pool Fires. *NISTIR 6546*. [online] Available at: <http://gisceu.net/PDF/U802.pdf>. [accessed 10 November 2012]
- Ma T. et al., 2004. Burning Rate of Liquid Fuel on Carpet (Porous Media). *Fire Technology* 40, 227-246. [online] Available at: <http://www.era.lib.ed.ac.uk/handle/1842/2635>. [accessed 15 January 2013]
- McDermott R.J. et al., 2010. *Fire Dynamics Simulator Version 6: Complex Geometry, Embedded Meshes, And Quality Assessment*. National Institute of Standards and Technology, Gaithersburg, Maryland, USA.
- McDermott R.J. et al., 2010. *Fire Dynamics Simulator (Version 5) Technical Reference Guide Volume 2: Verification*. NIST Special Publication 1018-5. [online] Available at: fire.nist.gov/bfrlpubs/fire07/PDF/f07048.pdf. [accessed 13 June 2012]
- McGrattan K. et al., 2007. Fire Dynamics Simulator (Version 5) User's Guide. *NIST Special Publication 1019-5*. [online] Available at: <http://fire.nist.gov/bfrlpubs/fire07/PDF/f07053.pdf>. [accessed 26 July 2012]
- McGrattan K. et al., 2012. Fire Dynamics Simulator (Version 6) User's Guide. *NIST Special Publication 1019-5*. VTT Technical Research Centre of Finland.
- Noah L. et al., 2004. Consequence Modelling Using the Fire Dynamics Simulator. *Journal of Hazardous Materials* 115, 149-154. [online] Available at: <http://www.sciencedirect.com/science/article/pii/S0304389404003607>. [accessed 26 July 2012]

- Petterson. N.M., 2002. *Assessing the Feasibility of Reducing Grid Resolution in FDS Field Modelling*. University of Canterbury
- Pope N.D. and Bailey C.G., 2006. Quantitative Comparison of FDS and Parametric Fire Curves with Post-Flashover Compartment Fire Test Data. *Fire Safety Journal* 41, 99-110. [online] Available at: <http://www.sciencedirect.com/science/article/pii/S0379711205001190>. [accessed 9 September 2012]
- Ronchi E., Pasquale C. and Nicola B., 2012. Reviewing Italian Fire Safety Codes for the Analysis of Road Tunnel Evacuations: Advantages and Limitations of Using Evacuation Models. *Safety Science*. [online] Available at: <http://www.sciencedirect.com/science/article/pii/S0925753512000793>. [accessed 9 September 2012]
- Shen T.S., Huang Y.H. and Chien S.W., 2006. Using Fire Dynamics Simulation (FDS) to Reconstruct an Arson Fire Scene. *Building and Environment* 43, 1036-1045. [online] Available at: <http://www.sciencedirect.com/science/article/pii/S0360132306003908>. [accessed 26 February 2013]
- Smardz P., 2006. *Validation of Fire Dynamics Simulator (FDS) for Forced and natural Convection Flows*. University of Ulster.
- Thunderhead Engineering. 2012. PyroSim User Manual. [online] Available at: <http://www.aeolus-consulting.be/downloads/pyrosimmanual.pdf>. [accessed 9 September 2012]
- Thunderhead Engineering. 2010. PyroSim Examples Guide. [online] Available at: <http://www.thunderheadeng.com/downloads/pyrosim/PyroSimExamples.pdf>. [accessed 26 July 2012]
- Versteeg H.K. and Malalasekera W., 2007. *An Introduction to Computational Fluid Dynamics The Finite Volume*. England: Pearson Prentice Hall.
- Viall R., 2008. *Fire Modelling in FLUENT*. Published final year project. Worcester Polytechnic Institute.
- Wang B., 2011. Comparative Research on FLUENT and FDS's Numerical Simulation of Smoke Spread in Subway Platform Fire. *Procedia Engineering* 26, 1065-1075. [online] Available at: <http://www.sciencedirect.com/science/article/pii/S1877705811051186>. [accessed 3 August 2012]
- Wang Y., Prateep C. and John L.R., 2010. Large Eddy Simulation of Fire Plumes. *Proceedings of the Combustion Institute* 33, 2473-2480. [online] Available at: <http://www.sciencedirect.com/science/article/pii/S1540748910002944>. [accessed 10 November 2012]
- Yang D. et al., 2010. Comparison of FDS Predictions by Different Combustion Models with Measured Data for Enclosure Fires. *Fire Safety Journal* 45, 298-313. [online] Available at: <http://www.sciencedirect.com/science/article/pii/S0379711210000494>. [accessed 13 June 2012]

- Yang P., Xun T. and Wang X., 2011. Experimental Study and Numerical Simulation for a Storehouse Fire Accident. *Building and Environment* 46, 1445-1490. [online] Available at: <http://www.sciencedirect.com/science/article/pii/S0360132311000229>. [accessed 26 February 2013]
- Yuan S. and Zhang J., 2008. Large eddy Simulation of Compartment Fire with Solid Combustibles. *Fire Safety Journal* 44, 349-362. [online] Available at: <http://www.sciencedirect.com/science/article/pii/S037971120800115X>. [accessed 10 November 2012]
- Zhanga W. et al., 2007. A Smoke Detector Activation Algorithm for Large Eddy Simulation Fire Modelling. *Fire Safety Journal* 43, 96-107. [online] Available at: <http://www.sciencedirect.com/science/article/pii/S0379711207000719>. [accessed 15 January 2013]

APPENDICES

APPENDIX A: UDF of Heat Flux Coding and Compilation Steps

The UDF coding of the heat flux is typed into Notepad and saved as extension .c, which is shown in below. The library code used is udf.h. The “define profile” is used to allow the boundary condition “wall” at the fire source to have a heat flux with ramp-up, steady state and decay period. Since the fire simulation is done up to 900 s, the equation of the HRR at ramp-up, steady state and decay period is a function of time, which the equation is obtained with variable time, t. With the UDF coding, the variable heat flux in the fire simulation by FLUENT can be mimic as same as FDS.

```
#include "udf.h"
```

```
DEFINE_PROFILE(wall_fire, thread, position)
```

```
{
```

```
    face_t f;
```

```
    real t= RP_Get_Real("flow-time");
```

```
    begin_f_loop(f, thread)
```

```
    {
```

```
        if (t<=300 && t>0)
```

```
            F_PROFILE(f, thread, position) = 1666.67*t;
```

```
        else if (t<=600 && t>300)
```

```
            F_PROFILE(f, thread, position) = 500000;
```

```
        else if (t>600)
```

```
            F_PROFILE(f, thread, position) = -1666.67*t + 1500000;
```

```

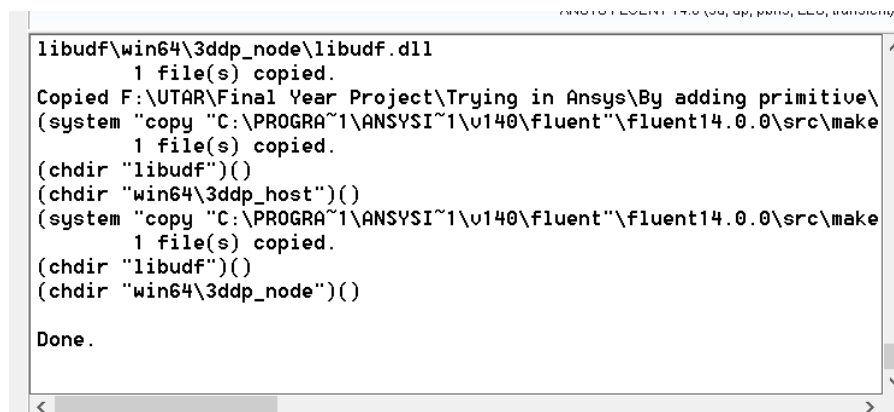
else
    F_PROFILE(f, thread, position) = 0;
}
end_f_loop(f, thread);

}

```

With Microsoft Visual Studio installed, there are several following steps need to be done so that the UDF can be compiled in the FLUENT.

- 1) Right click Computer, select Properties. Choose the Advanced system settings on the left tab.
- 2) In the pop-up menu, select Advanced tab. Click on Environment Variables.
- 3) Look into System variables, select Path and click edit.
- 4) Add the following into the Variable value, "C:\Program Files (x86)\Microsoft Visual Studio 10.0\Common7\Tools;C:\Program Files (x86)\Microsoft Visual Studio 10.0\VC\bin;C:\Program Files\ANSYS Inc\v 140\fluent\ntbin\win64" without the inverted comma.
- 5) After this, the UDF can be interpreted and compiled in the FLUENT itself. A successful compilation of UDF in FLUENT will show message as below.



```

libudf\win64\3ddp_node\libudf.dll
1 file(s) copied.
Copied F:\UTAR\Final Year Project\Trying in Ansys\By adding primitive\
(system "copy "C:\PROGRA~1\ANSYSI~1\v140\fluent"\fluent14.0.0\src\make
1 file(s) copied.
(chdir "libudf")()
(chdir "win64\3ddp_host")()
(system "copy "C:\PROGRA~1\ANSYSI~1\v140\fluent"\fluent14.0.0\src\make
1 file(s) copied.
(chdir "libudf")()
(chdir "win64\3ddp_node")()

Done.

```

APPENDIX B: UDF of Con-premixed Combustion Model

The UDF for the non-premixed combustion model is different from the UDF of heat flux model. In UDF for heat flux, the variation is on the heat flux while in non-premixed combustion model, the UDF is created to vary the velocity profile at the fire inlet. A pyramid shape of velocity profile is created for the fire source. First, the centroid of the fire source must be known. Then the equation of the velocity of the velocity inlet can be determined with respect to the location of the centroid. The velocity inlet is set to at peak of 0.5 ms^{-1} . The UDF of the non-premixed combustion model is as below:

```

/*A udf coding to create the flow velocity profile at z-direction*/
/*The centroid of the fire inlet is (-0.3 , -3.7)*/

#include "udf.h"                                /*library for user define function (UDF)*/

DEFINE_PROFILE(wall_fire, thread, position)      /*specify a profile
condition on a boundary face zone*/
{
    real wall_fire[ND_ND];
    real x,y;
    real t;
    face_t f;
    begin_f_loop(f, thread)
    {
        F_CENTROID(wall_fire, f, thread); /*store a boundary condition in
memory for a given face and thread*/
        x = wall_fire[0];
        y = wall_fire[1];
    }
}

```

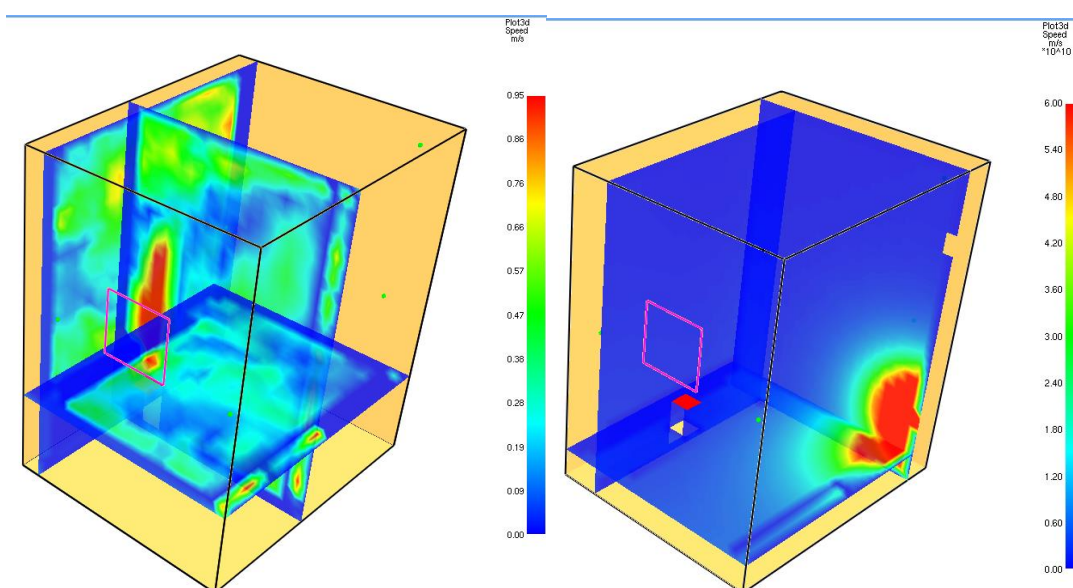
```
if (x < -0.3 && y < -3.7)
    F_PROFILE(f, thread, position) = 0.5 +
250*(x+0.3)*(x+0.3)*(x+0.3) + 250*(y+3.7)*(y+3.7)*(y+3.7);
else
    F_PROFILE(f, thread, position) = 0.5 -
250*(x+0.3)*(x+0.3)*(x+0.3) - 250*(y+3.7)*(y+3.7)*(y+3.7);
}
end_f_loop(f, thread);
}
```

With UDF, the velocity profile can be created. However, there is caution required in generating the UDF for velocity profile. The equation must be well known so that the precise velocity profile can be obtained. In this case for the non-premixed combustion model, there are so many unknown such as the velocity inlet value, fuel type, air fuel mixture ratio, exact velocity profile, etc. In generating this UDF, the velocity inlet value, fuel type, air fuel mixture ratio, exact velocity profile are not well known and all values are just assumed. The simulation results show that the output is not as expected as what defined earlier, which is discussed in the Chapter 4.

APPENDIX C: Analytical Step of Numerical Instability in FDS

In FDS, numerical instability occurs when the CFL condition is not satisfied. In solving equation, FDS adjust the timestep based on the velocity. Numerical instability occurs when the velocity calculated is too large which turns the timestep to a very small value, as close to zero.

To check the numerical instability in FDS, using SmokeView, the Plot3D of the last timestep is displayed. By moving the XYZ plane of the Plot3D, a region of large velocity can be spotted in the event of numerical instability. From the figure below, the left figure shows a normal Plot3D file without numerical instability. On the other hand, the right figure shows a case of numerical instability where the red region shows a sudden rise in velocity. The magnitude of the velocity displayed at high value, which is at $6 \times 10^{10} \text{ ms}^{-1}$, in this case, it is shown that numerical instability had occurred.



APPENDIX D: Comparison of Temperature Slice, Velocity Vector Slice and Temperature Isosurface of Mesh Resolution Study

The comparison of temperature slice, velocity vector slice and temperature isosurface of factor $D/\delta x$ at 2, 4 and 8 are shown in the table below from 300 s to 600 s at interval of 100 s. The simulation results are in the softcopy in CD.

APPENDIX E: Comparison of Temperature Slice and Temperature Isosurface of Meshing Method

The comparison of temperature slice and temperature isosurface of different meshing methods are as below from 300 s to 600 s at interval of 100 s. There are five methods to be compared, which are 1 mesh, 2 mesh, embedded with aligned mesh, embedded with unaligned mesh and surrounded mesh. Method (2 mesh) encountered numerical instability which there will be no result from 400 s onwards. The simulation results are in the softcopy in CD.

APPENDIX F: Comparison of Temperature Slice, Velocity Vector Slice and Temperature Isosurface between FDS and FLUENT

For the comparison between FDS and FLUENT, the temperature slice, temperature isosurface and velocity vector slice are compared from 300 s to 600 s at interval of 100 s. The simulation results are the amended input set where the convective heat flux from FDS had turned on and the fire source of FLUENT had adjusted to suit FDS input. The simulation results are in the softcopy in CD.

APPENDIX G: Comparison of Temperature Slice, Velocity Vector Slice and Temperature Isosurface of Various Geometry

The temperature slice, velocity vector slice and temperature isosurface are compared between 1 mesh method and embedded with aligned mesh method. There are few curvature geometries simulated, which are the cylindrical column, ramp (parallel and non-parallel to axis) and hemisphere structure. The simulation results are in the softcopy in CD.