

FIRE MODELING IN FLUENT

A Major Qualifying Project Report:

submitted to the Faculty

of the

WORCESTER POLYTECHNIC INSTITUTE

in partial fulfillment of the requirements for the

Degree of Bachelor of Science

by

---

**Robert Viall**

---

**Karl Wiegand**

Date: April 24, 2008

Approved:

---

Professor Grétar Tryggvason, Ph.D., Advisor

1. Fire modeling
2. FLUENT
3. CFD

This report represents the work of one or more WPI undergraduate students submitted to the faculty as evidence of completion of a degree requirement. WPI routinely publishes these reports on its web site without editorial or peer review.

## **Abstract**

In this project the possibility of using FLUENT, a commercial computational fluid dynamics software, for simple fire modeling is explored. FLUENT is widely used for numerous industrial fluid flow applications and could provide those familiar with the program a convenient way to run simple fire models. The team tested multiple methods for simulating a fire with various boundary conditions in FLUENT, including the built in combustion model and a more simplified approach utilizing a heat source.

## **Acknowledgements**

The team would like to thank all those who have helped us with this project over the past three terms. Special thanks to Adriana Hera of the WPI Computing and Communications Center for her help in getting us started with using both Gambit and FLUENT. The team would also like to thank our project advisor Professor Grétar Tryggvason, Ph.D. for his continued guidance and support throughout this project.

## **Executive Summary**

Fire modeling is an important and useful tool for research, fire reconstruction, and building design analysis. Since the 1960s, analytical fire models have been used to capture various aspects of a fire. These mathematical models are the basis for the more powerful computational models used today. Fire models can be categorized into either a zone or a field model. A zone model usually divides the compartment into two distinct zones, a hot upper gas layer and a cooler lower gas layer whereas a field model utilizes thousands or even millions of tiny control volumes upon which the governing equations can be applied. The first computer fire model was introduced in the United States in 1975 (Babrauskas).

Currently, computer fire models can be used to solve for gas and surface temperatures, species production, flow rates, and sprinkler and detector activation. In general, the effects of a fire can be modeled, but not the fire itself. There are a number of free and commercially available programs designed specifically for fire modeling as well as some computation fluid dynamics programs that can simulate certain aspects of the fire such as smoke flow. One of the most widely used fire modeling programs is the National Institute of Standards and Technology's (NIST) Fire Dynamics Simulator (FDS) which was developed in 2000 and is available online for free.

The purpose of this project was to explore the possibility of using FLUENT, a commercial computational fluid dynamics software, for simple fire modeling. FLUENT is widely used for numerous industrial fluid flow applications and could provide those familiar with the program a convenient way to run simple fire models. The team tested multiple methods for simulating a fire with various boundary conditions in FLUENT, including the combustion model and a more simplified approach using a source of heat.

Four of FLUENT's major solvers were used during this project including the combustion model, the energy equation, the P1 radiation model, and the k-epsilon turbulent flow model. FLUENT was designed to run combustion models for furnaces, boilers, process heaters, gas turbines, and rocket engines in order to predict flow field and mixing characteristics, temperature fields, species concentrations, and particulates and pollutants. The energy equation is used to account for heat transfer in a FLUENT model. Heat transfer can occur by three main methods: conduction, convection, and radiation. The radiation model was used to show the compartments interaction with its surroundings. The k-epsilon model is a two equation model that accounts for turbulent flows, heat transfer, and mass transfer in FLUENT.

In order to accomplish the project's objectives, the team created a typical sized residential compartment to be used as the basis for the simulations. Each vent was divided into two sections at a height of 1.5m above the floor in order to simulate the two way vent flow that would occur in an actual fire. The volume was meshed and imported into FLUENT where the boundary conditions and governing equations were set.

The first major step was to perform a grid refinement study to assure the accuracy of the CFD model by determining what size mesh will give enough accuracy for the model to be valid while minimizing simulation time. A test simulation was run using a number of different mesh sizes until the results of the simulation were not significantly different from the previous simulation with the more coarse mesh.

Two main approaches were used to model the fire in FLUENT; the program's combustion model and a simplified model using a source of heat. An adiabatic non-premixed combustion model was used in conjunction with the energy equation and the k-epsilon turbulent flow model. A vent in the center of the floor provided an inflow of Methane ( $\text{CH}_4$ ) gas at a rate of 1 m/s as the fuel source. For the second set of simulations, the fire was modeled as a source of heat through a circular vent in the floor. The fire was defined as an inflow with a temperature of 600K and a velocity of 5m/s. A number of different trials were run using this method with various boundary conditions; velocity inlets and outlets where air flowed through the lower part of the vents a rate of 0.5 m/s, and pressure inlets and outlets.

FLUENT's animate function was used to plot the results over a given time interval. The outputs of interest for most fire modeling applications are temperature, velocity, and pressure. Pressure was ignored in the majority of these simulations as a simplification. Three dimensional plots, along with graphs of temperature and velocity were used to view the data. The graphs showed the temperature and velocity at specific points in the compartment including the origin of the flame, the fire plume, the upper and lower portions of the vents, and the upper and lower layer in the compartment. In order for FLUENT to record data at specified points, each point was mapped with FLUENT's point generator.

The final step was to conduct validation exercises to check the accuracy of the model. In FLUENT, the fire size was indirectly input by providing a temperature and velocity, but since most fire models require the energy output of the fire to be directly input, it was necessary to calculate this by hand. The first law of thermodynamics was used to convert this into an energy output in kilowatts. CFAST, a widely used zone modeling program developed by NIST, was used as one method of checking

the model. The MQH upper layer correlation was also used to validate the model since it can be quickly calculated to roughly determine the upper layer temperatures.

## Results

The results of the simulations utilizing the adiabatic non-premixed combustion model showed that this method is not appropriate for modeling a fire in a standard compartment. This simulation produced temperatures that were significantly higher than a real fire. Maximum temperatures in the compartment reached over 2000K, which is much too hot for this type of fire. The problem is that the combustion model in FLEUNT is designed and most often used for modeling combustion engines; a quite different scenario than a non-premixed fire in a room.

The next set of simulations modeled the fire as a heat source with various boundary conditions. Since pressure plays an important role in the dynamics of a fire scenario as it controls smoke and air flow, pressure boundaries were tested. The result showed that two-way vent flow was not achieved. After 120s, the flow through both openings was out of the compartment. It was determined that by simply using a heat source to model the fire, there was no combustion which would have created a hydrostatic pressure difference.

The final model tested used the heat source model of the fire with the vents set as velocity inlets and outlets. The final model reached very high temperatures by the end of the simulation, with the majority of the upper portion of the room reaching 500K. From the contour plots, it was determined that a distinct hot upper layer had not formed like the team had expected. Probes were used to monitor the temperatures of the upper and lower layers so that the model could be compared to CFAST.

The CFAST model was run with a fire of 183 kW, and found that after the fire reached steady state, the interface height dropped down to 1.6m above the floor as opposed to our assumed value of 1.5m. The total upper layer temperature reached 150C° (423 K), with a lower layer temperature of 31.5C° (304.5K). For the MQH hand calculations, the group chose to use gypsum, aka sheetrock, as the wall material because it is a common material with known materials properties. The results of the MQH correlation gave the final temperature to be close to 400K which agrees with the CFAST model and suggests that the FLUENT model is over predicting the temperatures.

The room did appear to be getting very warm though and did not appear to be approaching a steady state. After investigating the details of the vents, the group discovered that the vents were not releasing heat from the compartment. After adjusting the boundary conditions using the P1 radiation model, the model produced temperatures in the upper layer between 400K and 420K. These results are

much closer to the results from CFAST and the hand calculations than before, however the lower layer temperatures are inaccurate.

## **Conclusions & Recommendations**

The overall objective of this project was to explore the possibility of using FLUENT for simulating simple fire models. It can be concluded that FLUENT is capable of performing these types of simulations particularly those aspects involving fluid flow such as smoke propagation. The results of this project show that FLUENT's adiabatic non-premixed combustion model is not appropriate for modeling diffusion flames in a compartment. It can be concluded that the fire can be model as a source of heat assuming a number of simplifications. There are numerous problems associated with running fire models in FLUENT. The three major problems are associated with:

- ❖ Fire size
- ❖ Boundary conditions
- ❖ Data output

Inputting the size of the fire presents a problem when it is being modeled simply as a source of heat. Most of the difficulties faced during this project were related to the boundary conditions of the model. This included the velocities at the vents and the temperature and velocity of the heat source. Once the simulations were run, it was difficult to extract useful data from the FLUENT. The animate feature was useful in viewing the overall conditions in the compartment over time, but it was difficult to get hard data for a given point location over the simulation period.

Some of the limitations of the model came about due to the simplifications made while others were due to the nature of the software and fire modeling in general. In order to address some of these limitations, the team recommends the following changes be made in future research:

- ❖ model a pool or coal fire
- ❖ vary the assumed neutral plane height
- ❖ specify a time dependent heat source
- ❖ test FLUENT's non-adiabatic premixed combustion model
- ❖ validate the model with FDS

In closing, this project has shown that FLUENT is capable of running fire related simulations to determine heat in the far field; however the use of a vent instead of a combustion process makes the near field data unreliable. As a result, those familiar with the program may find it beneficial to use FLUENT for simple fire models in certain situations instead of having to learn a program designed solely for fire modeling such as FDS.

# Table of Contents

Abstract.....	ii
Acknowledgements.....	iii
Executive Summary.....	iv
Results.....	vi
Conclusions & Recommendations .....	vii
List of Illustrations.....	x
List of Tables .....	xi
1 Introduction .....	1
2 Background .....	3
2.1 Limitations of fire modeling.....	4
2.2 What is FLUENT? .....	5
2.2.1 FLUENT solvers .....	5
2.2.2 Examples of FLUENT fire simulations .....	6
2.3 State-of-the-art Fire Modeling.....	8
3 Methodology.....	11
3.1 Gambit.....	11
3.2. FLUENT .....	12
3.2.1 Grid refinement .....	12
3.2.2 Combustion model .....	12
3.2.3 Heat source model .....	12
3.2.4 FLUENT Outputs .....	14
3.3 Validation .....	14
4 Results & Analysis .....	16
4.1 Grid refinement .....	16
4.2 Combustion model.....	18
4.3 Pressure model .....	19
4.4 Final Model .....	20
4.5 Validation .....	23
4.5.1 The First law of Thermodynamics .....	23
4.5.2 CFAST Validation.....	24
4.5.3 Hand Calculations .....	25



4.6 Revised Final Model .....	27
5 Conclusions .....	31
6 References .....	33
Appendix A: Grid refinement results .....	35

## List of Illustrations

Figure 1. Typical Zone Model .....	3
Figure 2. Example of field model control volumes .....	3
Figure 3. Smoke flow in tunnel using FLUENT .....	6
Figure 4. Modeling smoke flow in Budapest Sports Arena using FLUENT .....	7
Figure 5. Plot of temperature in FDS simulation .....	8
Figure 6. Principles behind positive pressure ventilation .....	10
Figure 7. Compartment dimensions .....	11
Figure 8. Creating compartment geometry in Gambit .....	11
Figure 9. Location of probes .....	14
Figure 10. Velocity plot using 0.075m grid spacing .....	17
Figure 11. Velocity plot using 0.050m grid spacing .....	17
Figure 12. Velocity plot using 0.040m grid spacing .....	18
Figure 13. Temperature plot of combustion test at t=10s.....	18
Figure 14. Temperature plot of combustion test at t=20s.....	19
Figure 15. Plot of velocity vectors for pressure test .....	19
Figure 16. Temperatures in the compartment at 300s.....	20
Figure 17. Velocity magnitudes in the compartment at 300s .....	20
Figure 18. Plot of temperature change over time for flame probes .....	21
Figure 19. Plot of temperature change over time for miscellaneous probes.....	21
Figure 20. Plot of change in velocity over time for flame probes.....	22
Figure 21. Plot of change in velocity over time for miscellaneous probes .....	23
Figure 22. Temperature output in Smokeview .....	24
Figure 23. CFAST temperature output.....	25
Figure 24. CFAST interface height output.....	25
Figure 25. MQH temperature change using Gypsum wallboard .....	26
Figure 26. Contour plot of temperature at t=300s .....	27
Figure 27. Contour plot of velocity at t=300s .....	28
Figure 28. Plume temperatures .....	28
Figure 29. Compartment temperatures.....	29
Figure 30. Compartment velocity magnitude .....	30
Figure 31. Plume velocity magnitude .....	30
Figure 33. Temperature contours for 0.10m grid .....	35
Figure 32. Temperature contours for 0.25m grid .....	35
Figure 36. Temperature contours for 0.04m grid .....	36
Figure 35. Temperature contours for 0.05m grid .....	36
Figure 34. Temperature contours for 0.075m grid .....	36

## List of Tables

Table 1. Grid refinement results .....	16
Table 2. MQP variables .....	26

# 1 Introduction

In recent years, there has been an increasing emphasis on fire safety. The tragedies of September 11, 2001 and the more recent Station Nightclub fire in Rhode Island have made people more aware of the importance of fire safety as it poses a large threat to both life and property. Advances in technology and research over the past few decades have significantly reduced the risk. For instance, the number of civilian fire deaths has decreased from 6,185 in 1985 to 3,675 in 2005. During the same time period, the number of fires was reduced from 2,371,000 to 1,602,000. However, fire still poses a significant risk that needs to be addressed as architects and engineers continue to push the limits and create taller, more unique structures. This trend has led to an increase in property damage from fires from \$7.324b to \$10.672b in that same twenty year span (NFPA Fire Incident Data Organization, 2006).

Fire modeling is an important and useful tool for research, fire reconstruction, and building design analysis. Since the 1960s, analytical fire models have been used to capture various aspects of a fire. These mathematical models were then combined to form more powerful computational models. Fire models can fall into one of two categories; either a zone or field model. A zone model usually divides the compartment into two distinct zones, a hot upper gas layer and a cooler lower gas layer. The governing equations are then applied to each zone in order to solve for the variables of interest. A field or CFD model breaks up the area into thousands or even millions of tiny control volumes upon which the governing equations can be applied. Advances in computer technology and fire research have vastly improved the capabilities of such models. The first computer fire model was introduced in the United States in 1975 (Babrauskas).

Currently, computer fire models can be used to solve for gas and surface temperatures, species production, flow rates, and sprinkler and detector activation. In general, the effects of a fire can be modeled, not the fire itself. Fire modeling programs only simulate a limited range of fire types, including gas burner, pool, solid, wall, ceiling, and spray fires. There are a number of free and commercially available programs designed specifically for fire modeling as well as some computation fluid dynamics programs that can simulate certain fire conditions such as smoke flow.

One of the most widely used programs for fire modeling is the National Institute of Standards and Technology's (NIST) Fire Dynamics Simulator (FDS). FDS was developed in 2000 and is available online free of charge. The program has been constantly maintained and revised since its development and represents the current scope and limitations of fire modeling. FDS has been used for various

applications including modeling smoke flow in multi-story buildings, tunnel fires, and different ventilation conditions (Fire Dynamics Simulator and Smokeview (FDS-SMV), 2008).

The purpose of this project was to explore the possibility of using FLUENT, a commercial computational fluid dynamics software, for simple fire modeling. FLUENT is widely used for numerous industrial fluid flow applications and could provide those familiar with the program a convenient way to run simple fire models. The team tested multiple methods for simulating a fire with various boundary conditions in FLUENT, including the combustion model and a more simplified approach using a source of heat.

## 2 Background

Analytical fire modeling was first developed in the 1960s and has been evolving ever since. Early analytical models focused on simulating a specific aspect of a fire scenario. As these mathematical models have developed, they have been combined to form powerful computational models.

There are two main categories of fire models: field models and zone models. Both types of models have their own strengths and weaknesses and are therefore best used in different scenarios. A zone model predicts the effects of a fire in a compartment by breaking up the compartment into zones of uniform conditions. Usually, two distinct zones are used; a hot upper smoke layer and a lower gas layer. In some cases, the fire plume and ceiling jet are also modeled as separate zones. Figure 1 shows the concept of a typical zone model. The conservation laws are applied to each zone and are solved for the quantities of interest. The strength of a zone model is that it is relatively simple.

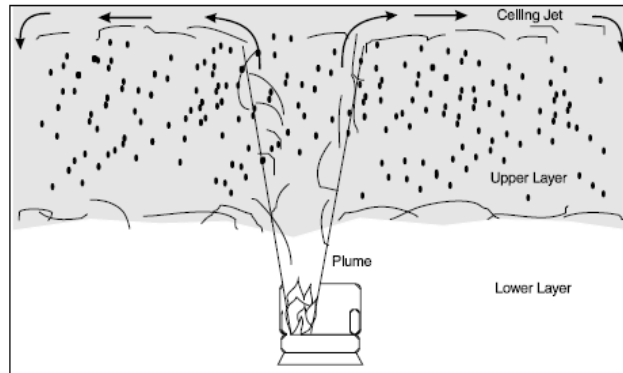


Figure 1. Typical Zone Model (nist.gov)

This reduces computation time and makes it easier for the user to analyze and truly understand the model. The drawbacks of such a model are that zone models are usually limited to a “standard” compartment with certain geometry, and cannot be used when there is no uniform smoke layer (Olenick and Carpenter 2003).

A field model does not break down a compartment into two large zones but many very small control volumes as shown in Figure 2. It is not unusual for a field model to be made up of thousands or millions of control volumes. The conservation equations are applied to each control volume resulting in more accurate solutions. Since the compartment no longer needs to be broken down into two uniform layers, field models can be applied to a wider range of problems than zone models. The downside of field models is that they require much more computational time and resources than zone models. They also usually require more input from the user (Olenick and Carpenter 2003).

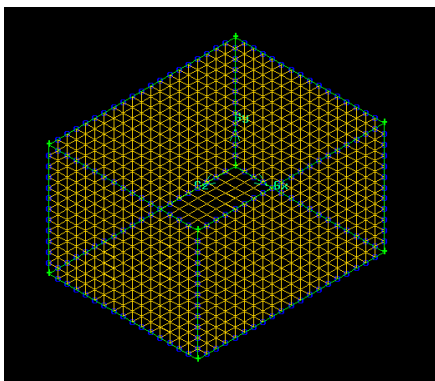


Figure 2. Example of field model control volumes

## 2.1 Limitations of fire modeling

Although computer fire models have become a powerful tool for fire protection engineers and researchers, there is still much needed improvement. The first computer fire model was introduced in the United States in 1975 (Babrauskas). Advances in fire science and technology have greatly improved the accuracy and scope of fire models. However, there is a common misconception that any fire models can be used for all types of situations. It is critical for users to understand the specific circumstances under which a model is valid. In general, the effects of a fire can be modeled, not the fire itself. Currently, gas and surface temperatures, species production, flow rates, and sprinkler and detector activation can be computed in using fire models. Fire models require the user to input information on the heat release rate of the fire, and species production. The user must reasonably choose the fire to include in the model, as it has been shown that the output is particularly sensitive to this input (Babrauskas).

The input parameters a user chooses for a fire are obtained from test data of controlled burns. If the fire or scenario the user is trying to model is not similar enough to the experimental test data which the user has chosen to define the model with, the model will not correctly represent the actual characteristics of a fire in that compartment. Because test data is not available for most situations, a level of inaccuracy must be expected and an understanding of fire behavior must be had for the output of the model to be of use. In many cases, the model may not be of any use at all except to show that the model is not valid for that particular scenario (Mowrer and Stroup 1998).

Fire modeling programs only simulate a limited amount of fire types, including gas burner fires, pool fires, solid fires, wall fires, ceiling fires, and spray fires. Gas burner fires use a programmed heat release rate (HRR) that are independent of environmental conditions. These fires do not always produce particularly accurate results, but are useful for finding problems in model parameters because they produce a constant HRR so if the results of the experiment do not go to a steady state then something else is wrong in the model. Pool fires consist of a liquid fuel with a horizontal surface of a specified area. The burning rate for these fires depends on the thermal feedback to the fuel surface. Solid fires are made up of a solid fuel of specified properties and geometry and, like pool fires, are dependent on heat feedback to the fuel surface. Wall fires and ceiling fires are self explanatory. Spray fires model a liquid fuel emitted under pressure through a puncture in a surface (Mowrer and Stroup 1998).

## 2.2 What is FLUENT?

FLUENT is a computational fluid dynamics (CFD) software. The first version of the FLUENT software was launched in October 1983 as collaboration between Sheffield University and a company called Creare Inc. FLUENT was written in the C computer language. The first version could involve both 2D and 3D structured grids using Cartesian or polar coordinates, steady-state flow, laminar or turbulent conditions, heat transfer, three-component combustion, a dispersed phase, and natural convection. FLUENT became very successful and in 1988 the FLUENT group at Creare broke away from the rest of the company establishing their own company, Fluent Inc. In August 1995, Fluent Inc. was acquired by Aavid Thermal Technologies Inc. FLUENT continued to improve and expand, buying Polyflow S.A. to gain POLYFLOW software code as well as improving their own code. In January 2000, Aavid Thermal Technologies was bought by Willis Stein & Partners. In May 2006, Fluent Inc. was bought by ANSYS Inc., the company that owns it today. The following sections discuss the models incorporated into FLUENT, particularly those related to modeling fire, as well as some documented examples in which FLUENT was used for fire related simulations.

### 2.2.1 FLUENT solvers

Four of FLUENT's major solvers were used during this project including the combustion model, the energy equation, the k-epsilon turbulent flow model, and the P1 radiation model. FLUENT, though mostly used to model fluid flows, was also designed to be used for various combustion applications. FLUENT was designed to run combustion models for furnaces, boilers, process heaters, gas turbines, and rocket engines to predict flow field and mixing characteristics, temperature fields, species concentrations, and particulates and pollutants. Combustion modeling in FLUENT is governed by four transport equations; mass, momentum (turbulence), energy, and chemical species, using models for combustion, pollutant, dispersed phase, and radiative heat transfer.

The energy equation, shown as Eq.1, is used to account for heat transfer in a FLUENT model. Heat transfer can occur by three main methods: conduction, convection, and radiation. Physical models involving only conduction and/or convection are the simplest, while buoyancy-driven flow or natural convection, and radiation models are more complex (FLUENT 6.3 User's Guide 13.2).

$$\frac{\partial}{\partial t}(\rho E) + \nabla * (\vec{v}(\rho E + p)) = \nabla * (k_{eff} \nabla T - \sum_j h_j \vec{J}_j + (\bar{\tau}_{eff} * \vec{v})) + S_h \quad [1]$$



The k-epsilon model is a two equation model that accounts for turbulent flows, heat transfer, and mass transfer in FLUENT. There are three types of k-epsilon models; standard, renormalization group theory (RNG), and realizable k-epsilon models. All three models have similar forms with two transport equations; however they differ in their methods of calculating turbulent viscosity, the turbulent Prandtl numbers that governs turbulent diffusion, and the generation and destruction of terms in the epsilon equation. The standard model equations, Equations 2 & 3, are the least complex of the three equation sets and can only be used for fully turbulent flow. The other two equation sets build off of the standard model and use fewer approximations to have a much broader range in accuracy and usability. However, because these two equation sets are more complex to calculate, they can significantly increase the run time of the model (FLUENT 6.3 User's Guide, 12.4).

$$\frac{\partial}{\partial t}(\rho k) + \frac{\partial}{\partial x_i}(\rho k u_i) = \frac{\partial}{\partial x_j} \left[ \left( \mu + \frac{\mu_t}{\sigma_k} \right) \frac{\partial k}{\partial x_j} \right] + G_k + G_b - \rho \epsilon - Y_M + S_k \quad [2]$$

$$\frac{\partial}{\partial t}(\rho \epsilon) + \frac{\partial}{\partial x_i}(\rho \epsilon u_i) = \frac{\partial}{\partial x_j} \left[ \left( \mu + \frac{\mu_t}{\sigma_\epsilon} \right) \frac{\partial \epsilon}{\partial x_j} \right] + C_{1\epsilon} \frac{\epsilon}{k} (G_k + C_{3\epsilon} G_b) - C_{2\epsilon \rho} \frac{\epsilon^2}{k} + S_\epsilon \quad [3]$$

The P1 radiation model is a simple radiation model that allows for temperatures on the exterior of the model to be accounted for. With the P1 model turned activated, the exterior temperature can be set to ambient (300K). This allows the heat inside the model to radiate out into the exterior as it would if the window and door were actually open. The P1 model is relatively simple for FLUENT to solve so it does not significantly increase the run time.

### 2.2.2 Examples of FLUENT fire simulations

FLUENT has been successfully used to model fire scenarios in a number of cases. In one instance, FLUENT was used to model smoke development and spread in an automotive tunnel. The objective of the project was to model smoke abatement in the Memorial Tunnel in West Virginia. Dealing with smoke from a fire is an important criterion in tunnel design. Use of a computer model is necessary in these situations, because it is extremely difficult to conduct

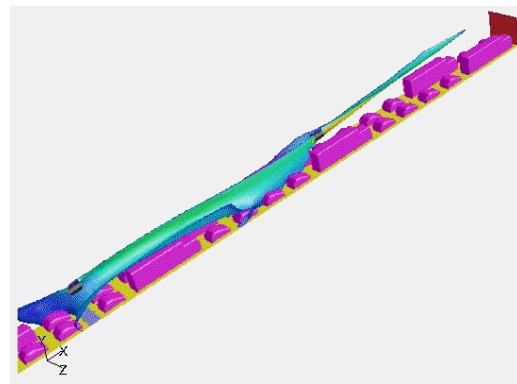
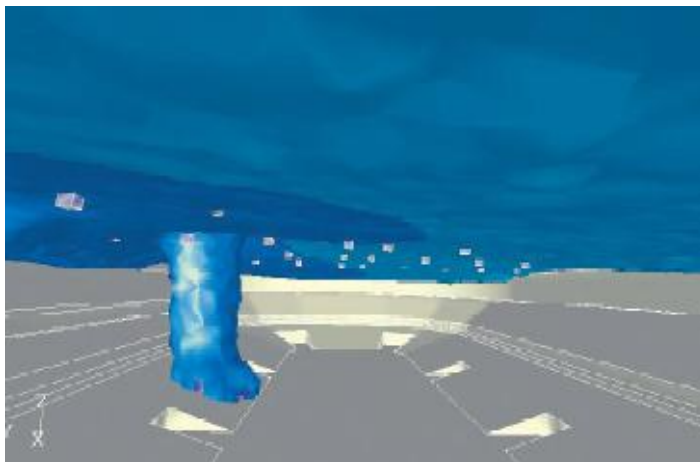


Figure 3. Smoke flow in tunnel using FLUENT (fluent.com)

tests. The FLUENT model was validated by comparing the results with experimental data. The results were comparable to the actual smoke spread in the tunnel. Figure 3 shows the smoke spread through the tunnel as simulated in FLUENT.

In a similar way, FLUENT was used to test various fire protection scenarios for the Frankfurt Airport. Analysis was performed on the airport's natural ventilation strategy under fire conditions. This was accomplished by simulating static pressure on the outside of the airport for different wind conditions. The results of the model can be used to determine the positioning of smoke detectors as well as test the effectiveness of the ventilation system. This is critical in ensuring that sufficient egress time is provided in the case of a fire.

One of the best documented uses of FLUENT in modeling a fire scenario involved the Budapest Sports Arena in Hungary. On December 15, 1999, a fire ravaged through the symbolic Budapest Sports Arena burning it to the ground and leaving a strong impact on the surrounding area. The arena was



**Figure 4. Modeling smoke flow in Budapest Sports Arena using FLUENT (fluent.com)**

quickly replaced with a design emphasizing safety. The design of the new arena incorporated a system to automatically open doors and close air vents as well as an exhaust system that would turn on in the event of a fire. FLUENT was used to model smoke flow in the arena in the event of a fire. In the early stages of a fire, hot smoke and toxic gases rise up to the ceiling and begin to form a smoke layer. This smoke layer slowly descends toward the floor as

more smoke is produced. It is critical that the ventilation system be designed to vent the smoke out fast enough to allow the occupants to escape before the smoke layer descends to head level. If the smoke descends too far, visibility would be greatly reduced and occupants would be exposed to hot toxic gases. In order to model the propagation of smoke throughout the arena, the fire was simulated by a source of heat and carbon dioxide. To account for a growing fire, the rate of heat and carbon dioxide production was modeled quadratically. Simulations were conducted for fires in various locations in order to fully analyze the design for different scenarios. Figure 4, shows the FLUENT model of smoke flow in the arena with the smoke being represented in blue. The results of the model concluded that the

ventilation system was adequate and would provide those seated in the upper grandstand a minimum of ten minutes to escape. The temperature of the smoke was also monitored to evaluate the potential for structural damage. From the simulation, it was determined that the air temperature would be approximately 100°C and would not pose a risk.

### 2.3 State-of-the-art Fire Modeling

Currently, the most widely used programs for fire modeling are those designed specifically for that purpose. These programs are widely available and are frequently updated as more research is completed. This section discusses one of the most widely used fire modeling programs and its limitations as it is a good representation of state-of-the-art fire modeling. Fire Dynamics Simulator (FDS) was publically released by the National Institute of Standards and Technology (NIST) in February 2000. The program is widely used because it is available online for free. FDS is a field model that solves a form of the Navier Stokes equations for low-speed, thermally driven flow. Since 2000, the program has undergone numerous revisions and upgrades. The most current edition is FDS Version 5, released in October 2007. NIST also developed Smokeview, a program that accompanies FDS in order to visualize the numerical output of FDS (Fire Dynamics Simulator and Smokeview (FDS-SMV), 2008). The next section will discuss the ways in which this program has been applied to various fire scenarios.

FDS has been used for countless fire applications. One of these applications focused on determining the height of the smoke layer in a compartment. Like FLUENT, FDS is good at modeling smoke layers. In a simulation published in the Tamkang Journal of Science and Engineering, FDS was used to determine if any of several types of ventilation systems would be able to adequately ventilate a 2 or 5mW fire. The objective was to keep the smoke layer greater than 10 feet above the upper level of the four story complex that was built to the 1988 California Building Code. Figure 5 shows the temperature output of the model.

It was determined that none of the proposed ventilation systems could adequately

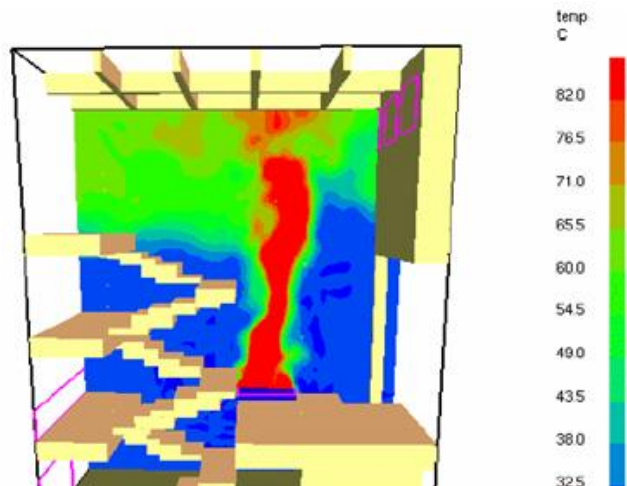


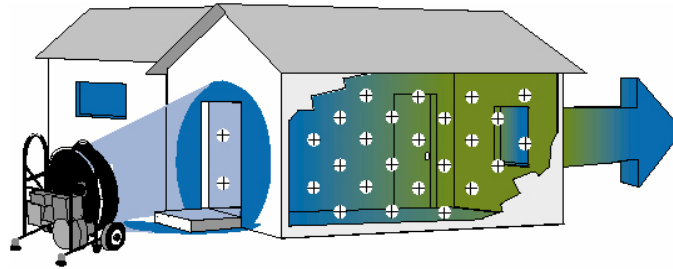
Figure 5. Plot of temperature in FDS simulation (Chang 2003)

ventilate the fire. The model showed that this was due to the high velocity of the fire plume. In this example, the plume was releasing smoke at 750 to 1000 feet per minute, which effectively “bounced” off the ceiling causing a lower smoke layer that even a large exhaust was unable to counter. It was also determined by calculations outside of FDS that external winds would probably cause the smoke layer to be even lower than it was shown in the model, something FDS could not simulate. This yet again shows the limits of the current fire modeling programs mentioned in the limitations section above (Chang 2003).

FDS can also be used to simulate tunnel fires. The program was used to model a fire in the L.H.La Fontaine tunnel in Montreal, Canada in order to test the smoke management system. It was used to make sure the road upstream of the fire was free of smoke with as little CO<sub>2</sub> as possible, with little or no backlayering in the tunnel. Backlayering occurs when the exhaust system does not have adequate ventilation capacity and the smoke moves toward an exit to leave the tunnel instead of towards the exhaust system. In this case, the fire was assumed to be a car fire which was modeled with a 15mW propane pool fire with an area of 3m<sup>2</sup>. It was determined that only a 300m section of the tunnel was necessary to input into the model because the temperature variance and CO<sub>2</sub> concentrations were low enough at the end of the 300m section to assume that the rest of the tunnel would not be affected much by the fire. Using only a section of an area to model a complex problem when possible is a good way of decreasing runtime, increasing grid refinement (accuracy), and decreasing cost associated with a project. It was determined that the ventilation system worked well when the upper side vents were closed and ventilation rate was high (Bounagui 2004).

FDS has also been used to test various affects on fires that help firefighters in fire scenarios. One of these tests involved the use of positive pressure ventilation (PPV) on fires. PPV is a common tactic used by fire departments to ventilate structures during and after a fire. In most cases, this helps the firefighters by increasing visibility and forcing heat away from the firefighters as they extinguish the flames and salvage the building. However, in some cases PPV can add oxygen to the fire and lead to fire growth. Figure 6 illustrates how PPV works in a building. FDS has been used by NIST to simulate fires in a “standard” colonial house to determine when PPV should and should not be used and where fans should be placed for maximum efficiency. The model showed that in the case being studied, the addition of a PPV would create tenable conditions in the house for firefighters to work in that would not have been present under normal ventilation conditions. Though this modeling experiment was performed by NIST and Homeland Security and the fire used in the model was tested in a combustion

chamber, there are still a number of uncertainties involved in this model. This is a helpful tool because it shows that in general PPV should be able to be used in this type of scenario, but not necessarily in all cases (Kerber 2006).



**Figure 6. Principles behind positive pressure ventilation (Kerber 2006)**

### 3 Methodology

This chapter details the methods used to test the capabilities of FLUENT in modeling a fire in a typical compartment. In any CFD simulation, the first step is to model the geometry of the compartment. Gambit was used to create and mesh the geometry of the room. The mesh file was then imported into FLUENT where the modeling equations and boundary conditions are set. Once the model was setup, a grid refinement exercise was carried out. After the preliminary setup and grid refinement was complete, the actual simulations were run using different approaches for modeling the fire. The following sections describe these steps in more detail.

#### 3.1 Gambit

A typical sized residential or small commercial compartment was used in all of the simulations because it provided a relatively simple basis for the simulations. The compartment consisted of a doorway and one window as illustrated in Figure 7. A three dimensional model of the compartment was created in Gambit. This

was accomplished by creating a series of volumes that were then united to form a single volume. Figure 8 shows the volumes that were created in Gambit. In all, six volumes were created; two for the each of the door and window, one for the fire vent, and one for the walls of the room. It was necessary to

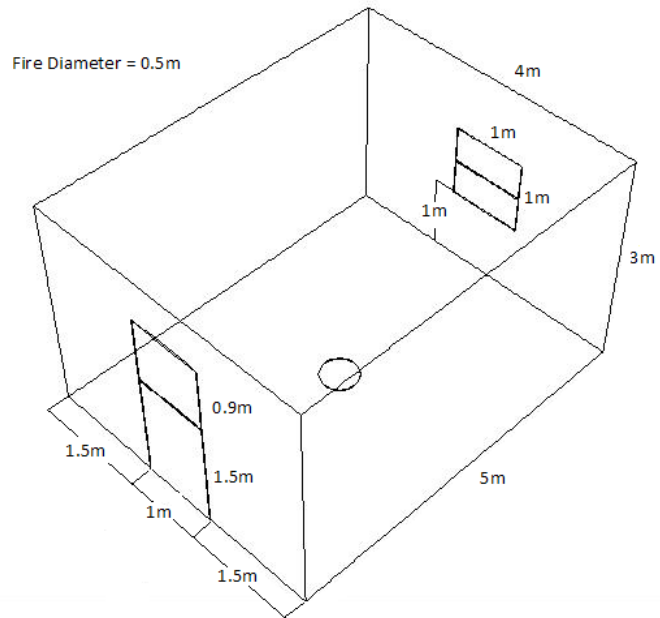


Figure 7. Compartment dimensions

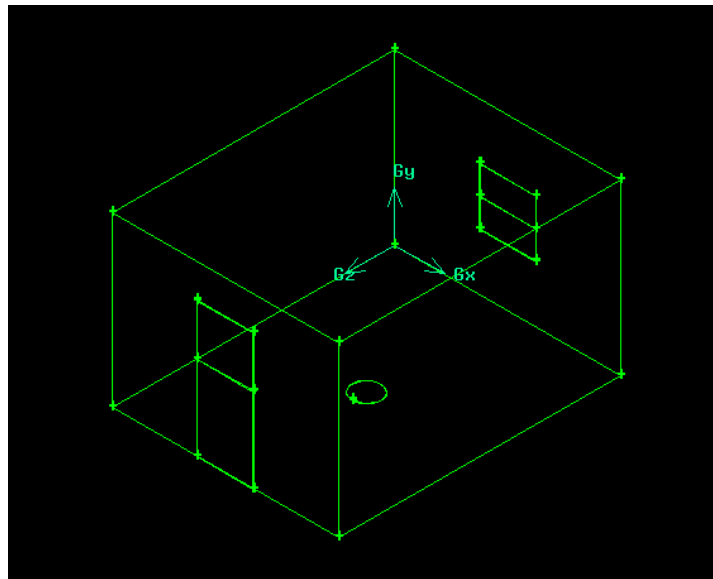


Figure 8. Creating compartment geometry in Gambit

construct the vents with separate volumes because the inflows and outflows can only be defined for an entire face. So in order to have an inflow through just a window as opposed to an entire wall, a separate volume was created. Each vent also had to be created as two separate vents in order to simulate two way vent flow. In an actual fire, there is usually an outflow through the upper part of a vertical vent and an inflow through the lower part. Next, the volume was meshed with a Tet/Hybrid Hex Core grid.

## **3.2. FLUENT**

Once the mesh file was imported from Gambit into FLUENT, the boundary conditions and governing equations were set. This step encompassed the majority of the work involved in this project. The next few sections describe the methods used to perform the grid refinement study and assess the different approaches to model fire in FLUENT.

### **3.2.1 Grid refinement**

As discussed previously in the background chapter, grid refinement plays a critical role in assuring the accuracy of a CFD model. The purpose of the grid refinement is to determine what size mesh will give enough accuracy for the model to valid while minimizing simulation time. Once the model was set up, a test simulation was run using a number of different mesh sizes. For each test case, the model was re-meshed in Gambit and imported into FLUENT. The results of each trial were then compared. The mesh size was reduced until the results of the simulation did not show any significant changes from those run with the more coarse mesh.

### **3.2.2 Combustion model**

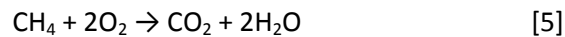
To test the ability of the combustion model to simulate a room fire, FLUENT's non-premixed combustion model was implemented. In addition, the energy equation and the k-epsilon turbulent flow model were used. In order to simplify the model and achieve a greater understanding of what was going on, the model was assumed to be adiabatic. The vent in the center of the floor provided an inflow of Methane ( $\text{CH}_4$ ) gas at a rate of 1 m/s. Air flowed through the lower portions of the door and window at a rate of 0.5 m/s. The top sections of the door and window were outflows.

### **3.2.3 Heat source model**

For these simulations, instead of using the combustion model and a fuel source, the fire is modeled as a source of heat through a circular vent in the floor. The energy equation and the viscous flow model were activated for these simulations. In a fire scenario, the flow is turbulent and therefore the k-epsilon

model was used. The boundary conditions are set in FLUENT. This includes inflow and outflow through the vents as well as the initial conditions. The upper portions were defined as inlet flows while the lower portions were outlet flows. A neutral plane of 1.5m was used. The neutral plane is the imaginary plane that separates inflow and outflow. The flow through the door and window was set at 300K with a velocity of 0 m/s. The fire was set as an inflow with a temperature of 600K with a velocity of 5m/s. Another simulation was run using the P1 radiation model to allow heat to be released into the exterior. Once the model was turned on the vents were defined as a path to the exterior. The exterior temperature was then set to 300 degrees Kelvin which was the group's default ambient temperature.

Because FLUENT is modeling a heat vent and not combustion, the air entrainment into the compartment is different than it should be. Combustion involves chemical reactions with the oxygen around it pulling air into the fire and releasing hot gases that travel upwards. For a methane fire the reaction for complete combustion is:



Entrainment in a combustion model is stronger than that of a vent. In combustion, air is pulled in to replace the air that was heated or burnt in the fire. Once the fire is large enough to require more air than is in the room around it, air is pulled in from the surrounding area. In the FLUENT "fire" a vent is being used to simulate the heat that would result due to combustion. Vents also cause convection, however due to the fact that air is being brought in through the fire vent whereas in actual combustion all air would be brought in through the window and door vents, there is less of a draw on the window and door vents. In order to counter balance this, the group created an "artificial entrainment" process to mimic the cold airflow that would be coming in from the ambient exterior, see below. This equation is a simplification of the entrainment, but will be fairly accurate as long as the fire does not go into a condition where it would smother itself; which a non-combustion model would not be able to represent anyway.

$$A_f * V_f = A_{iv} * V_{iv} \quad [6]$$

Where:

$A_f$ = area of fire

$V_f$ =velocity of fire

$A_{iv}$ = total area of vents below neutral plane

$V_{iv}$ = velocity of vents below neutral plane (each is the same)



### 3.2.4 FLUENT Outputs

In order to save and make use of data obtained throughout a FLUENT simulation, the animate function must be used. Animate can save any data FLUENT produces during a simulation over a period of intervals related to either the iterations or the time steps used in the model. When using animate, it is important to make sure that each desired type of data is set up to record correctly or the run will be wasted as unspecified output data cannot be obtained from a run that has already been performed. In fire modeling, temperature, velocity, and pressure are the most important outputs to monitor, but since pressure was ignored in this model as a simplification, only temperature and velocity outputs were recorded. FLUENT has a variety of ways to view the data that it outputs. Three dimensional view models, and temperature and velocity graphs were used to view the data for this project. The three dimensional view models showed the general air current movements and temperature distribution over the entire simulation. Contour plots of a certain plane were used to view changes in temperature for simulations where the distribution was too complex. Without using these slice views the flame path could not accurately be seen due to the cool ambient air that was introduced to the system .

The graphs showed the temperature and velocity at specific points in the compartment including the origin of the flame, the flame plume, and the upper and lower doorway and window. In order for FLUENT to record data at specified points, each point must be mapped with FLEUNT's point generator, see Figure 9.

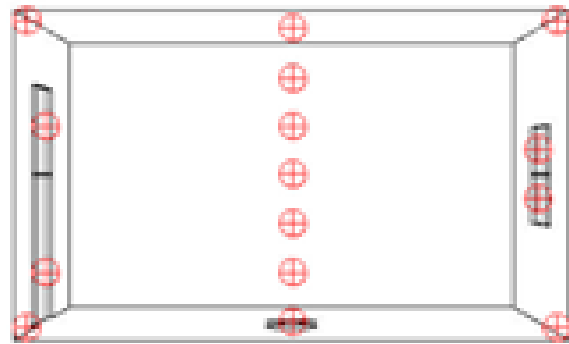


Figure 9. Location of probes

### 3.3 Validation

Obtaining results from the model does not necessarily mean that any useful information has been obtained. As was stated in the background, there are many simplifications to be dealt with. Therefore, the final step of this project was to conduct exercises in validation to test the accuracy of the model by determining the attributes of the fire using different methods. Even if the validation tests show similar trends in output, it does not mean that model is necessarily correct or not. However, the more similar the test results are from the different methods, the more likely the model is accurate. One way of showing accuracy in models is to reconstruct the model using another modeling program. In order for any of these validations to be carried out, the energy output of the fire must be known. In

FLUENT, fire size was indirectly input by providing a temperature and velocity. The first law of thermodynamics, shown below, must be used to convert this into a kilowatt output.

$$Q - W = (\Delta h + \Delta \dot{K}E + \Delta PE) * \dot{m} \quad [7]$$

CFAST, a widely used zone modeling program developed by NIST, was used as one method of checking the model. CFAST is appropriate for this scenario because it is a quick running program and is excellent at modeling cube shaped compartments with average size, well ventilated fires. Another way of performing validations is through hand calculations such as the MQH upper layer correlation. The MQH upper layer correlation is a fairly simple, compared to computer fire models, temperature calculation for the upper layer of a fire created by McCaffrey, Quintiere, and Harkleroad. MQH is not as accurate as the other models, but it gives another way to validate the results (Walton and Thomas, 2002). The best form of validation would be a controlled burn. A controlled burn under the same conditions as the model would show the actual temperatures and velocities associated with the fire as it burns. However, a controlled burn was not feasible for validation purposes of this project.

## 4 Results & Analysis

The chapter discusses the outcome of the project in regards to the methods stated in the previous chapter. Before any of the test simulations could be run, the results of the grid refinement were needed. After the appropriate mesh size was determined, simulations were run utilizing FLEUNT's combustion model. A simpler approach of modeling the fire as a heat source from the floor was taken after the combustion model proved to be inappropriate for this type of fire model. Various boundary conditions were tested such as pressure and velocity. The best simulation resulted from using a heat source in the center of the compartment along with the use of a simulated neutral plane height and corresponding inflow and outflow. This chapter ends with a discussion on the results of the validation of the model that was created for this project.

### 4.1 Grid refinement

Grid Spacing (m)	# mesh volumes	Results
0.250	16,290	Results did not appear correct, too coarse
0.100	216,845	Different results from 0.25m mesh
0.075	335,641	Velocity plume was too tall
<b>0.050</b>	<b>1,347,754</b>	<b>Appropriate resolution</b>
0.040	1,607,633	Similar results to 0.05m grid spacing
0.025	-----	Computer froze, simulation took too long

Table 1. Grid refinement results

As discussed previously in this report, grid refinement is an important part of accurately running a CFD simulation. To accomplish this task, different values for the grid spacing were used and the results were compared. The coarsest grid used was 0.25m with finer grids of 0.10, 0.075, 0.05, 0.04, and 0.025m. As the grid spacing decreased, the number of nodes in the model increased rapidly. Plots of the temperature and velocity contours were analyzed for each simulation and compared to each other. It was determined that the results of the simulation did not show significant changes when then grid

spacing was reduced from 0.05m to 0.04m. Contour plots of velocity are shown in Figures 10-12 using the 0.075, 0.05, and 0.04m grid spacing. Notice how the simulation with the 0.075m grid spacing shows a more advanced stage of the velocity plume as the plume is taller than in the other simulations. A coarser grid has a shorter computation time and should be used if possible. Therefore, a grid spacing of 0.05m was chosen for our test simulations because the plots for the 0.05m and 0.04m grid sizes were very similar. A summary of the results of the grid refinement study are shown in Table 1. Note that the fine 0.025m mesh did not run successfully. Plots of the temperature contours for the different simulations are shown in Appendix A. The temperature plots were all very similar and therefore could not be used to differentiate the appropriate grid size.

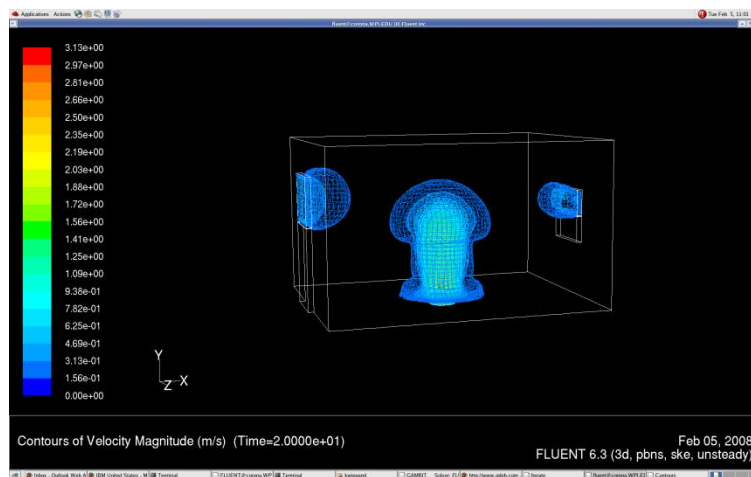


Figure 10. Velocity plot using 0.075m grid spacing

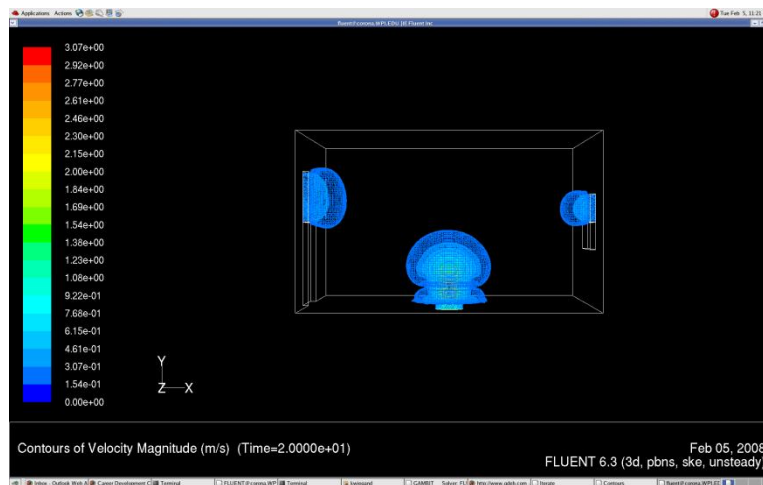


Figure 11. Velocity plot using 0.050m grid spacing

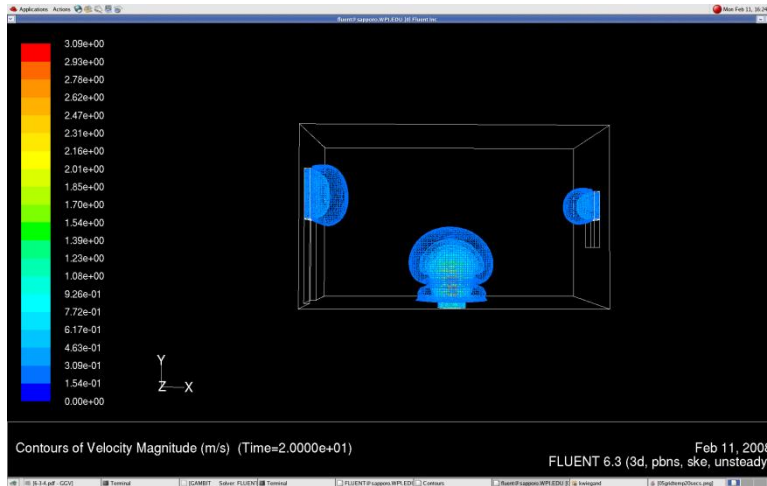


Figure 12. Velocity plot using 0.040m grid spacing

## 4.2 Combustion model

Use of the combustion model in FLEUNT had the most potential for success since it is directly related to fire scenarios. However, the results of the simulations utilizing this combustion model showed that this method was unsuccessful in accurately modeling a fire in a standard compartment. This simulation produced temperatures that were significantly higher than a real fire. Figures 13 and 14 show contour plots of the temperature distributions within the compartment at 10s and 20s respectively. The plots show that the maximum temperatures in the compartment reached over 2000K, which is much too hot this early on in a fire. The problem is that the combustion model in FLEUNT is designed for and most often used in modeling combustion engines as mentioned in section 2.2.1 of this report; a quite different scenario than a non-premixed fire in a room.

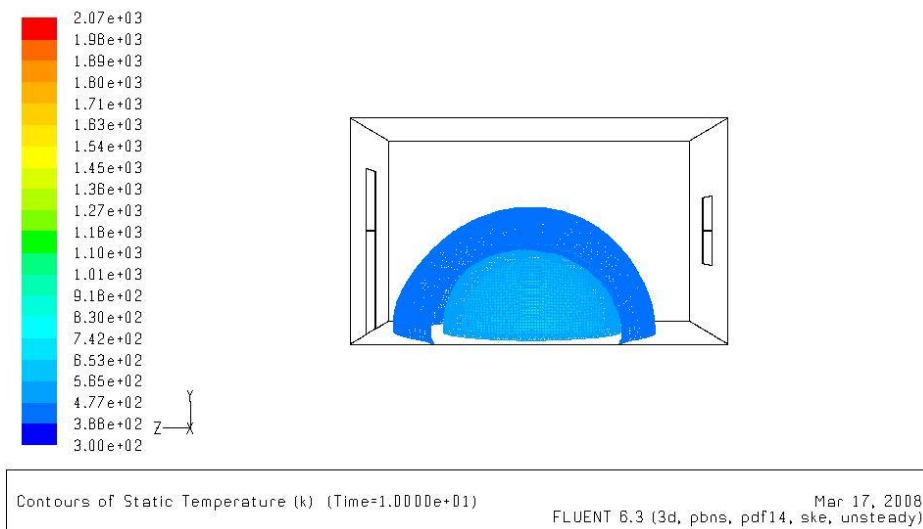


Figure 13. Temperature plot of combustion test at t=10s

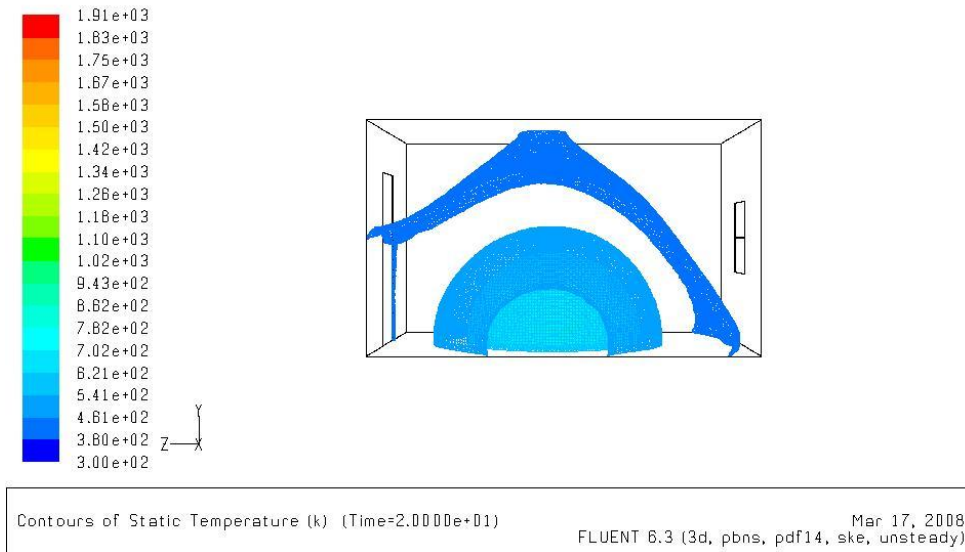


Figure 14. Temperature plot of combustion test at t=20s

### 4.3 Pressure model

Pressure plays an important role in the dynamics of a fire scenario as it controls smoke and air flow. The hydrostatic pressure difference across horizontal flow vents leads to an outflow in the upper region of the vent and an inflow in the lower region. This vent flow affects the air entrainment and ultimately the fire size. In order to simulate this phenomenon, the model was created using pressure vents for the door and window. The result was that two-way vent flow was not achieved. As shown in Figure 15, after 120s the flow through both openings was out of the compartment. The blue seen in the figure are the numerous vectors showing the direction of the flow out of the compartment. Note that the vectors are not shown to scale. In analyzing these results, it was determined that this was due to the way the fire was modeled. By simply using a heat source to model the fire, there was no combustion which would have created a hydrostatic pressure difference.



Figure 15. Plot of velocity vectors for pressure test

## 4.4 Final Model

This section details the results of the final model in which a heat source was used. Figure 16 is a contour plot of temperature at a time of 300 seconds. The highest temperatures are centered in the compartment which is expected since that is where the heat source is located. From this plot, it can be seen that the final model reached very high temperatures by the end of the simulation with the majority of the upper portion of the room reaching 500K. Figure 17 shows the velocity magnitude at the same point in time.



Figure 16. Temperatures in the compartment at 300s

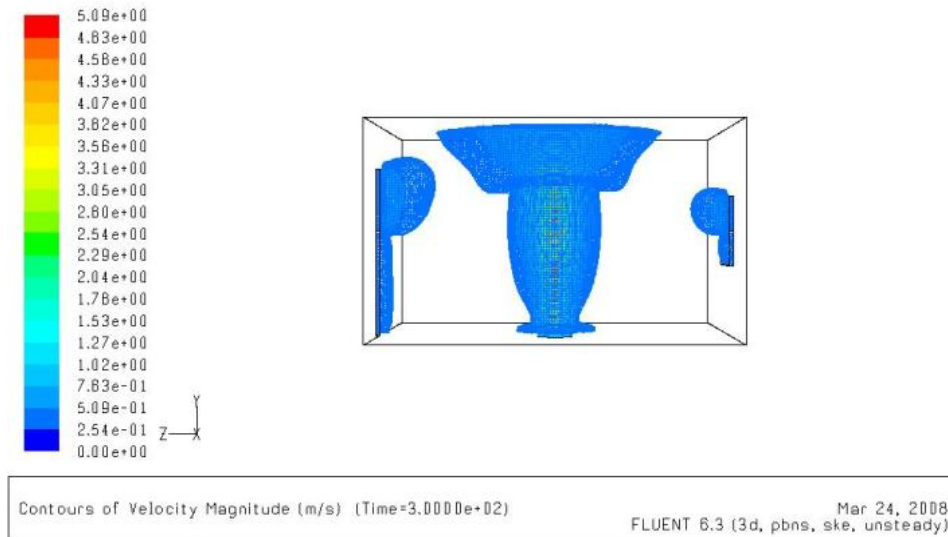


Figure 17. Velocity magnitudes in the compartment at 300s

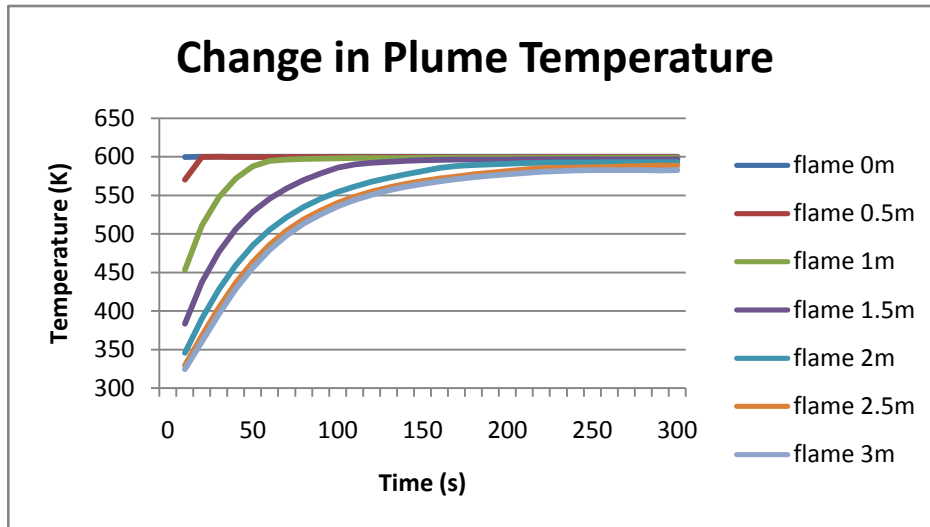


Figure 18. Plot of temperature change over time for flame probes

Figure 18 illustrates the change in temperature recorded by the probes located in the fire plume at different heights from the floor. As expected, the higher probes took longer to reach the steady state of around 600K than did the lower probes. This data shows the general characteristics of the flame, letting the user know how quickly the flame is growing in height as well as temperature. This flame appears to be growing at a normal rate of speed for a fire of this size.

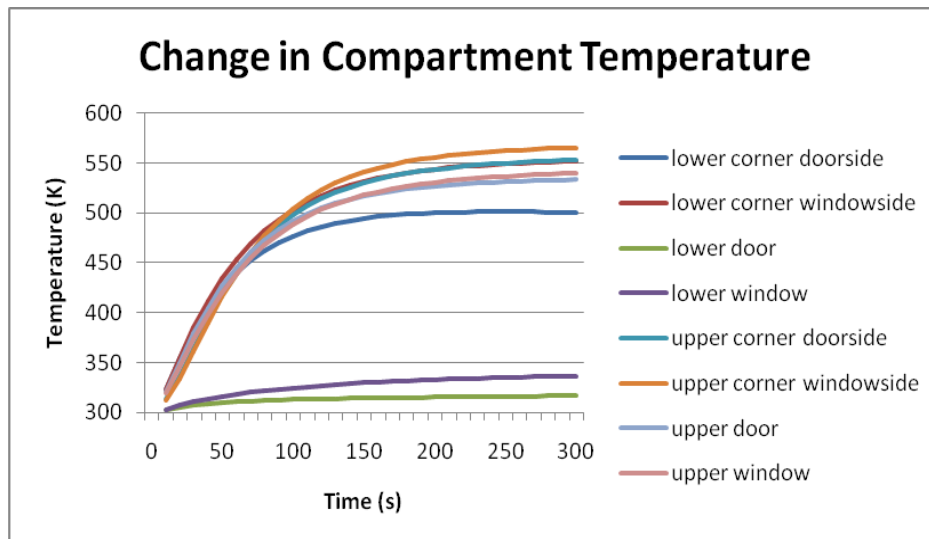


Figure 19. Plot of temperature change over time for miscellaneous probes



The temperature in the rest of the compartment was also monitored by probes as seen in Figure 19. These probes plot data in the upper layer (upper corner on each side), the lower layer (lower corner on each side), and the upper and lower vents in the compartment. With these probes, the temperatures of the upper and lower layer in general can show the temperature trends in the model. They will also be useful later on to compare the FLUENT model to the CFAST zone model and the MQH equations for validation. The flame patterns show that the compartment is coming to a steady state temperature in the 500K to 550K range. This seems to be bit high for a fire of this size, but could be plausible. The lack of a lower layer though is somewhat questionable as the only temperature probes that are exhibiting lower layer characteristics are the lower areas of the vent which is due to the 300K inlet temperatures that was defined as one of the boundary conditions.

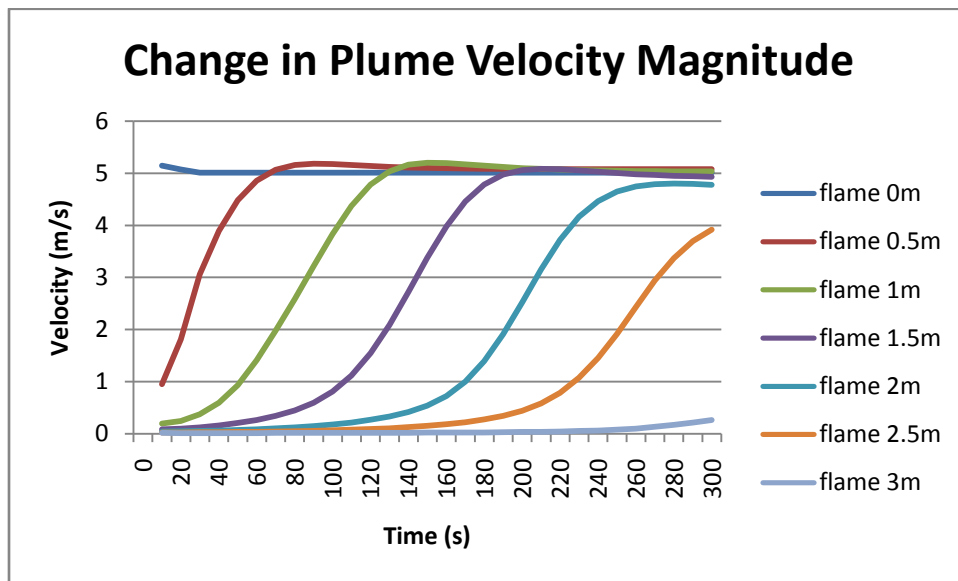


Figure 20. Plot of change in velocity over time for flame probes

The velocity in the fire plume, plotted in Figure 20, helps to show the characteristics of the fire plume. Like Figure 18, it shows the general flame height as the fire gets progressively larger. When the velocity hits five meters per second, the flame inlet speed, the flame has fully moved into that area of the compartment.

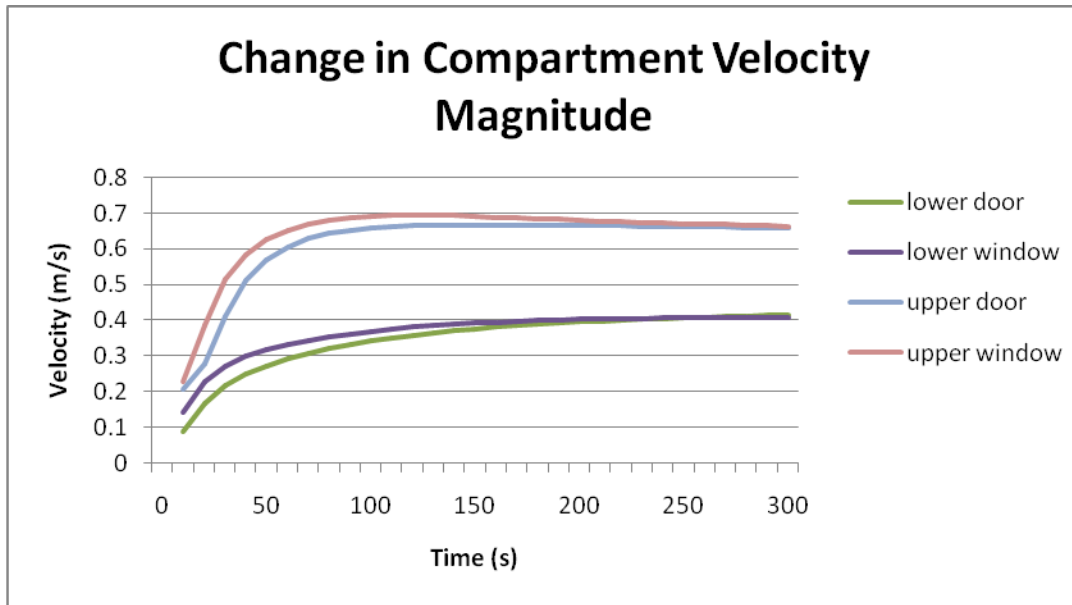


Figure 21. Plot of change in velocity over time for miscellaneous probes

The velocity probes placed in the vents are important because they can show whether or not the fire is properly entraining air from the outside. The probes in the upper and lower layer do not have much value because there is not much information to compare them to, so they were left out of the graph to avoid unnecessary confusion. The velocities at both the outlets and the inlets appear to be fairly low considering the fire size and our estimated entrainment.

## 4.5 Validation

This section of the results discusses the team's findings from the validation efforts. First, the first law of thermodynamics was used to determine the energy output of the fire. This was a necessary input into both CFAST and the hand calculations. The results of the CFAST simulation are given and compared with the output from FLUENT. Lastly, the findings from the hand calculations are detailed.

### 4.5.1 The First law of Thermodynamics

In order to use other fire modeling programs and equations for validating the FLUENT model, the size of the fire is needed. Equation 1, described earlier in the methodology chapter was used to find the energy output of the fire,  $Q$ . This was done by first assuming  $W=0$  since it is known that no work is done on the fire. Changes in kinetic and potential energy,  $\Delta KE$  and  $\Delta PE$ , are also both zero. The change in enthalpy is given by:

$$\Delta h = c_p * \Delta T = 315.3 \text{ kJ/kg} \quad [8]$$

The mass flow rate was found with the following equation:

$$m = A * V * \dot{\rho} = 0.58 \text{ kg/s} \quad [9]$$

$$\dot{Q} + 0 \text{ kW} = \left( 315.3 \frac{\text{kJ}}{\text{kg}} + 0 \frac{\text{kJ}}{\text{kg}} + 0 \frac{\text{kJ}}{\text{kg}} \right) * 0.58 \frac{\text{kg}}{\text{s}} \quad [10]$$

$$Q = 182.94 \text{ kW} \quad [11]$$

From these calculations, the fire size was determined to be around 183 kW which is an average size fire for the size compartment being used in these simulations. Using this result, validation could then be completed.

#### 4.5.2 CFAST Validation

A CFAST model was run with a fire of 183 kW in a compartment with the exact same dimensions as the main test compartment. After the fire reached steady state, the interface height went down to 1.6 meters above the floor. This shows that our 1.5m estimate was very reasonable for a fire of this magnitude. The total upper layer temperature reached 150°C (423 K), with a lower layer temperature of 31.5°C (304.5K). The CFAST temperature data is plotted in Figure 23.

Figure 22 shows the screenshot of the CFAST visual output in Smokeview of the temperatures from the fire once it reaches steady state. The fire plume is shown in red as well as the hot upper layer. It is also possible to view flow through the door in this figure. The upper layer has not descended over the plume and the fire is not outside of reasonable flame temperature for a fire of this size.

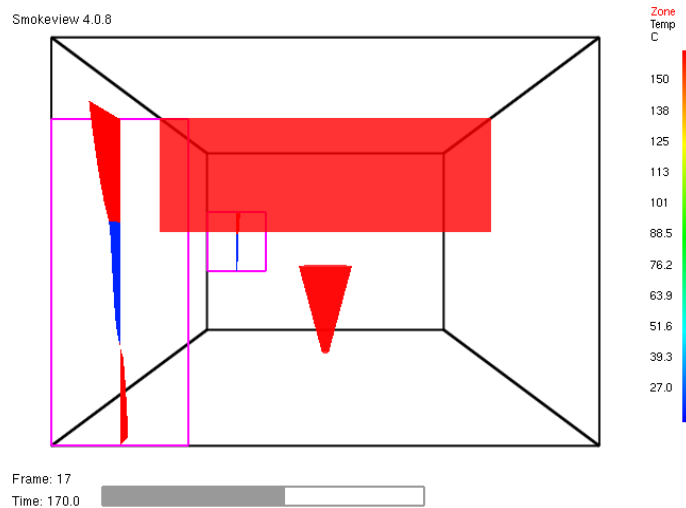


Figure 22. Temperature output in Smokeview

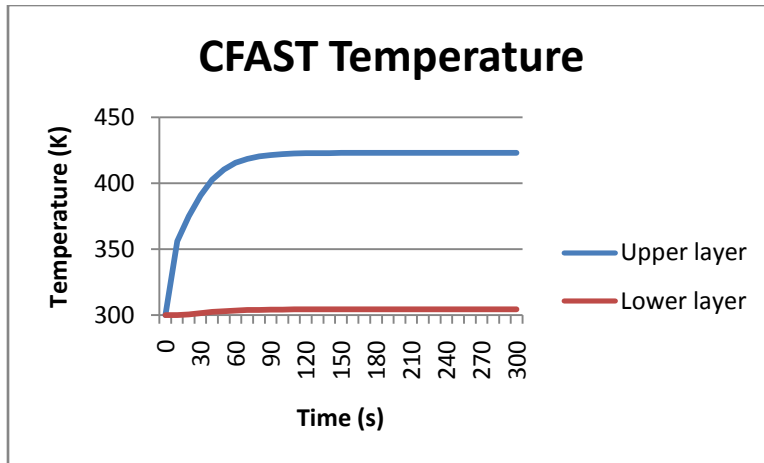


Figure 23. CFAST temperature output

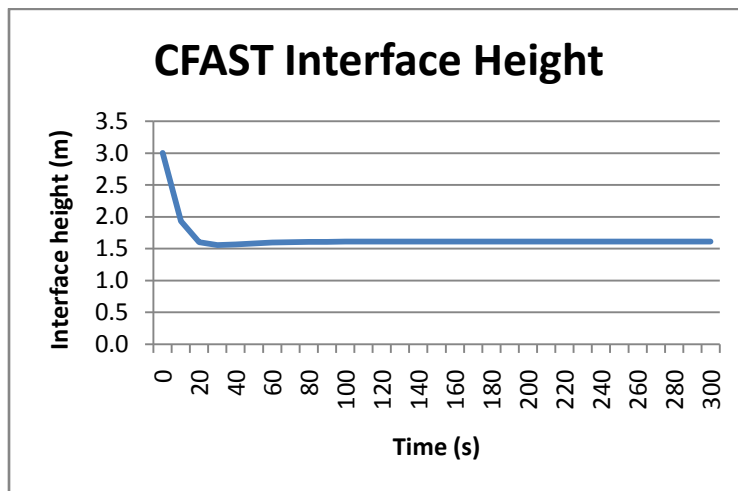


Figure 24. CFAST interface height output

The interface height plays an important role in the dynamics of the fire as it dictates vent flow and can limit combustion. It is important to note that the CFAST simulation predicted an interface height of around 1.6m for the majority of the simulation as illustrated in Figure 24. Remember that in the FLUENT simulations, an artificial neutral plane height was assumed to be at a height of 1.5m. The similarity of these results validates the value used as the neutral plane height.

### 4.5.3 Hand Calculations

MQH is a method of roughly calculating temperature change in the upper layer of a fire “by hand”. Once the variables below are obtained it can be done by simply plugging them into the equation below. The relevant variables and their values are shown in Table 2.

$$\Delta T = 480 * \left( \frac{Q}{\sqrt{g} * c_p * T_{\infty} * \rho_{\infty} * A_0 * \sqrt{H_0}} \right)^2 * \left( \frac{h_k * A_T}{\sqrt{g} * c_p * \rho_{\infty} * A_0 * \sqrt{H_0}} \right)^{-1} \quad [12]$$

Variable	Description	Value
Q	Energy (kW)	182.9
g	Gravity (m/s <sup>2</sup> )	9.8
c <sub>p</sub>	Specific Heat of Air (kJ/kg*K)	1.05
ρ <sub>∞</sub>	Density Ambient (kg/m <sup>3</sup> )	1.2
T <sub>∞</sub>	Temperature Ambient (K)	300
A <sub>T</sub>	Area of Enclosing Compartment Surfaces (m <sup>2</sup> )	90.6
ρ	Density Gypsum (kg/m <sup>3</sup> )	1440
k	Thermal Conductivity of Gypsum (kW/kg*K)	0.00048
c	Specific Heat of Gypsum (kJ/kg*K)	0.84
A <sub>o</sub>	Area of Openings (m <sup>2</sup> )	3.4
H <sub>o</sub>	Average Height of Openings (m)	2.2
t	Time (s)	0-300
h <sub>k</sub>	Effective Heat Transfer Coefficient (kW/m*K)	(k*ρ*c/t) <sup>0.5</sup>
ΔT	Change in Temperature (K)	?

Table 2. MQP variables

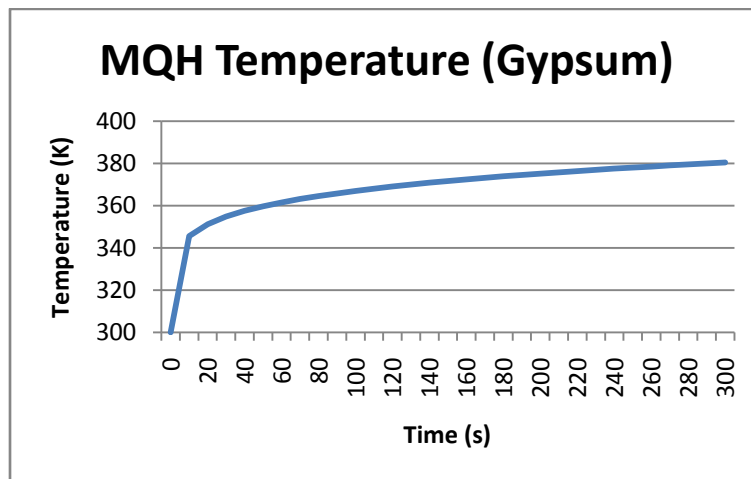


Figure 25. MQH temperature change using Gypsum wallboard

Though the group did not specify a material for the boundaries for either of the computer models, gypsum wallboard was used to define the walls for the MQH upper layer correlation. MQH requires the material properties of a wall in the equation. The group chose gypsum, aka sheetrock, because it is a common material for walls, is readily available, and the exact materials properties are known. Figure 25 shows a plot of the temperature output from the MQH calculations. The use of

different wall materials will affect the results. For instance, if fur was used then the final MQH temperature at five minutes would be 404K while brick would result in a final temperature of 375K. Thus the MQH correlation, even with a variety of materials, gives the final temperature to be close to 400K. This agrees with the CFAST calculation which suggests that the FLUENT calculation is most likely too high.

#### 4.6 Revised Final Model

As the flame appeared to be fairly correct, the group did not believe this was the problem. The room did appear to be getting very warm though and did not appear to be approaching a steady state. This being said, the group decided that the room was not correctly ventilating heat. After investigating the details of the vents, the group discovered that the vents were indeed not performing correctly. There was no release of heat from the compartment, so all of the heat that had been pumped in through the fire remained in the compartment and made it significantly hotter than expected. From the FLUENT User's Guide, it was discovered that the P1 radiation model would activate an ambient exterior and cause the heat to be released. For the simulation, artificial entrainment was also added to push air in from the ambient exterior to simulate entrainment that would have been present if there was actual combustion.

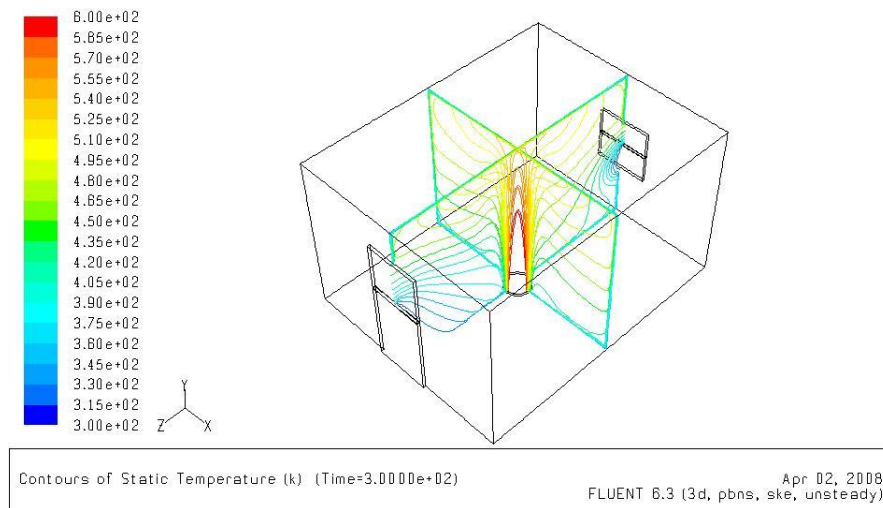


Figure 26. Contour plot of temperature at t=300s

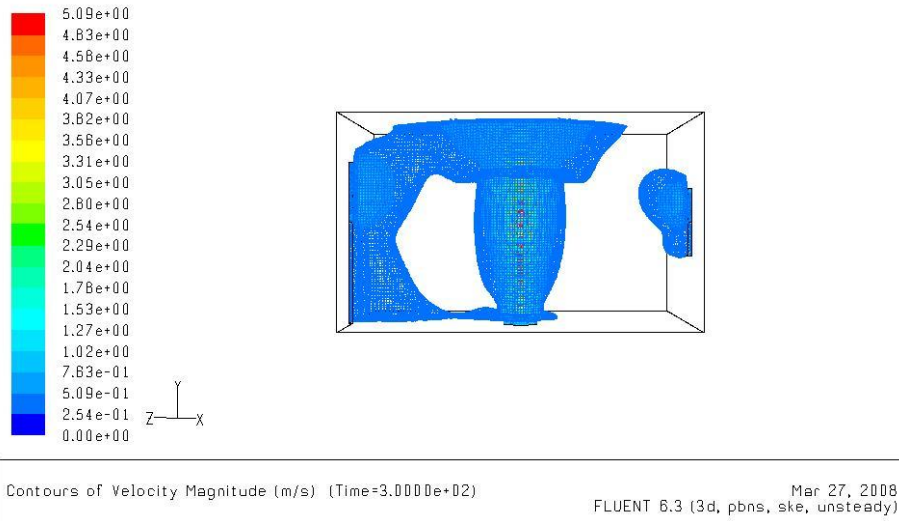


Figure 27. Contour plot of velocity at t=300s

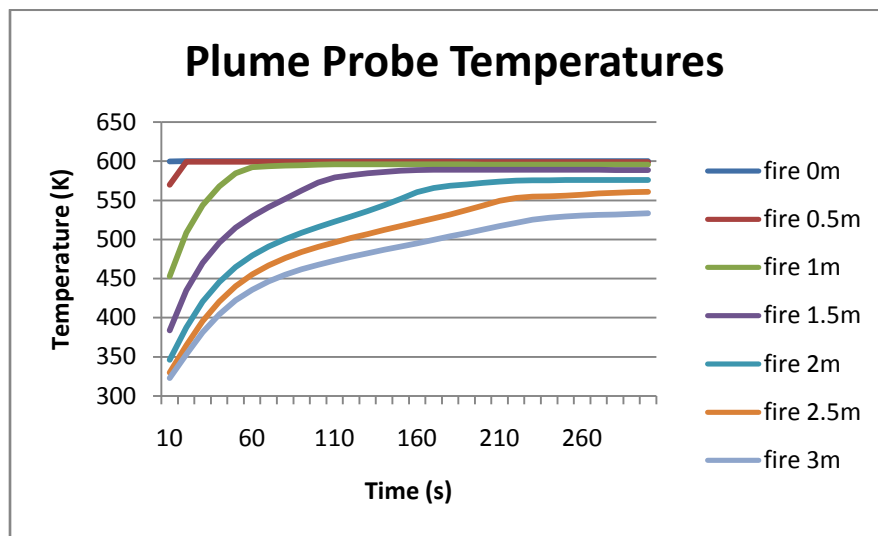


Figure 28. Plume temperatures

Figure 26 shows an isometric view of the temperature contour plots for the two center planes in the compartment. Figure 27 shows the velocity magnitude at the end of the simulation. The fire plume can be clearly seen in the center of the figure. With the modified P1 radiation model, the temperature of the fire did not rise as quickly as shown in Figure 28. This is logical because with an ambient exterior there will be heat loss whereas in the previous experiment no heat was released.

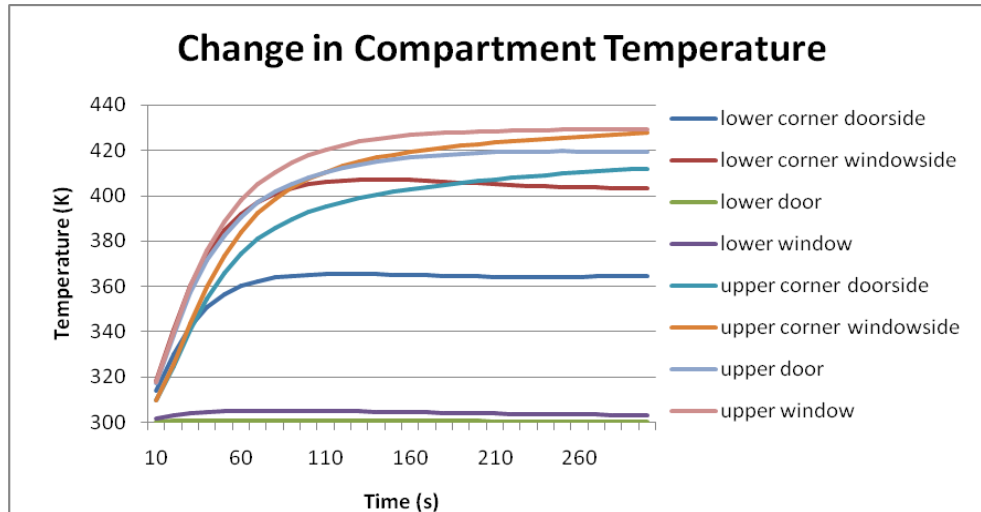


Figure 29. Compartment temperatures

The output from the compartment probes look much more agreeable with what was found in CFAST and MQH. The compartment came to a steady state when the temperature stopped rising, which it had not done in the previous model. With a well ventilated fire of this constant size in a room this size, it is expected that the fire will reach steady state. The temperatures recorded at the lower window and the lower door were extremely low once again since the vents were set as inlets of ambient air. The lower layer temperature was also fairly high. As can be seen in Figure 29, the temperature was higher on the window side of the room than on the door side. This shows that it was the cool air from the door that was keeping that lower area cold and was not a result of the fire. This makes sense because the fire was simulated with a vent which would radiate heat in all directions, more so than the combustion process would have. The group believes that the FLUENT model is a good method of determining far field because in this region the behaviors of a flame and a vent are more similar because in this range they are both basically just a heat source. However, for near field approximations the FLUENT model is not that useful as there are differences between the heat release properties of a vent and a fire in the area where combustion takes place.

As compared to the previous simulation, the fire plume velocities decreased as illustrated in Figures 30 & 31. This shows that the fire did not move up through the room as quickly as it had in the previous model as expected. The lower door and window velocities were set to 0.5 m/s from equation 5 in the methodology. The upper door and window velocities show that the fire reaches a steady state at about 150 seconds.



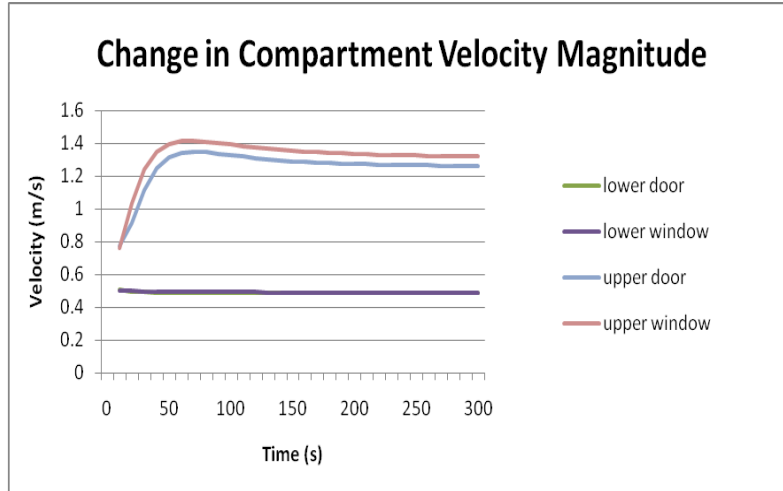


Figure 30. Compartment velocity magnitude

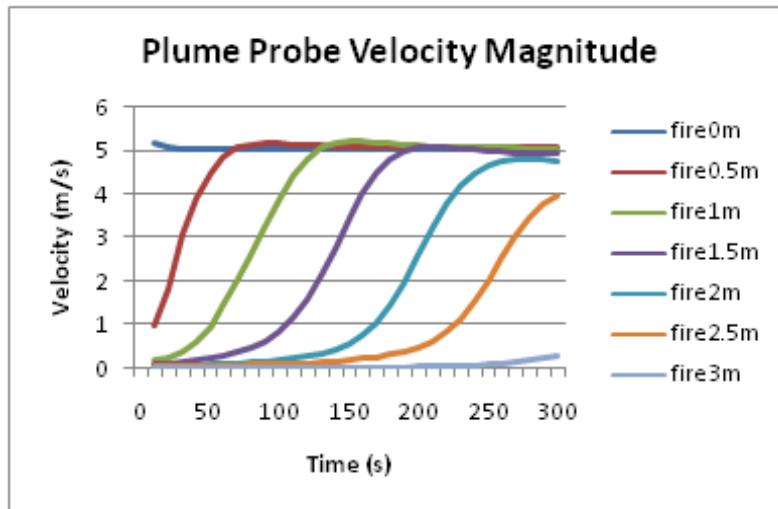


Figure 31. Plume velocity magnitude

## 5 Conclusions

There are a number of conclusions that can be drawn from this project. The overall objective of this project was to explore the possibility of using FLUENT for simulating simple fire models. It can be concluded that FLUENT is capable of performing these types of simulations particularly those aspects involving fluid flow such as smoke propagation. This is also documented in the examples given in the background chapter of this report. The results of this project show that FLEUNT's adiabatic non-premixed combustion model is not appropriate for modeling diffusion flames in a structure. It can be concluded that the fire can be model as a source of heat assuming a number of simplifications. There are numerous problems associated with running fire models in FLUENT. The three major problems are associated with:

- ❖ Fire size
- ❖ Boundary conditions
- ❖ Data output

Inputting the size of the fire presents a problem when it is being modeled simply as a source of heat. Most of the difficulties faced during this project were related to the boundary conditions of the model. This included the velocities at the vents and the temperature and velocity of the heat source. Once the simulations were run, it was difficult to extract useful data from the FLUENT. The animate feature was useful in viewing the overall conditions in the compartment over time, but it was difficult to get hard data for a given point location over the simulation period.

The simulations run for this project were highly simplified and therefore could have significant drawbacks. The specific limitations of our team's model includes the lack of:

- ❖ a limiting factor for pyrolysis
- ❖ air entrainment into the plume
- ❖ pressure
- ❖ an unsteady heat release rate (a growing fire)

Some of the limitations came about due to the simplifications assumed for this project while others are due to the nature of the software and fire modeling in general. In order to address some of these limitations, the team has compiled a number of recommendations on how to further investigate the use of FLUENT for fire modeling and possibly improve the accuracy of the model. These recommendations include:

- ❖ modeling a pool or coal fire
- ❖ varying the assumed neutral plane height
- ❖ specifying a time dependent heat source

- ❖ testing FLUENT's non-adiabatic premixed combustion model
- ❖ validating the model with FDS

In closing, this project has shown that FLUENT is capable of running fire related simulations in order to determine temperatures in the far field. However, the use of a heat source vent instead of a combustion process makes the near field data unreliable. As a result, those familiar with the program may find it beneficial to use FLUENT for simple fire models in certain situations instead of having to learn a program designed solely for fire modeling such as FDS.

## 6 References

- "Fire Dynamics Simulator and Smokeview (FDS-SMV)." National Institute of Standards and Technology. <http://fire.nist.gov/fds/> (accessed April 14, 2008).
- "The U.S. Fire Problem." NFPA Fire Incident Data Organization. <http://www.nfpa.org/itemDetail.asp?categoryID=953&itemID=23033&URL=Research%20&%20Reports/Fire%20statistics/The%20U.S.%20fire%20problem> (accessed March 11, 2008).
- "Fire and Smoke Modeling." FLUENT, Inc. <http://www.fluent.com/solutions/examples/x195.htm> (accessed January 27, 2008).
- FLUENT 6.3 User's Guide*. ANSYS, Inc.
- "Tunnel Fire Modeling." FLUENT, Inc. <http://www.fluent.com/solutions/examples/x148.htm> (accessed January 27, 2008).
- Babrauskas, Dr V. "Fire Modeling: An Introduction for Attorneys." Fire Science and Technology, Inc. [http://www.doctorfire.com/mod\\_test.html](http://www.doctorfire.com/mod_test.html) (accessed January 27, 2008).
- Bounagui, A., A. Kashef, and N. Benichou. "CFD Simulation of the Fire Dynamics for a Section of a Tunnel in the Event of a Fire." *CFD 2004, the 12th Annual Conference of the Computational Fluid Dynamics Society of Canada* (May 9, 2004).
- Chang, Cheng-hsin, David Banks, and Robert N. Meroney. "Computational Fluid Dynamics Simulation of the Progress of Fire Smoke in Large Space, Building Atria." *Tamkang Journal of Science and Engineering* 6, no. 3 (2003): 151-157.
- Kerber, Stephen. *Evaluation of the Ability of Fire Dynamic Simulator to Simulate Positive Pressure Ventilation in the Laboratory and Practical Scenarios*. Gaithersburg, MD: National Institute of Standards and Technology, 2006, <http://www.fire.nist.gov/bfrlpubs/fire06/PDF/f06065.pdf>.
- Kristóf, Dr G., Lohász, Máté, Régert, Tamás and Bodor, Péter. "Fire Scenarios in the Budapest Sports Arena." <http://www.fluent.com/about/news/newsletters/03v12i2/s7.htm> (accessed January 27, 2008).
- McGrattan, Kevin. *Fire Modeling: Where are we? Where are we Going?*. Gaithersburg, Maryland: National Institute of Standards and Technology, .
- McGrattan, Kevin, Bryan Klein, Simo Hostikka, and Jason Floyd. *Fire Dynamics Simulator (Version 5) User's Guide* 2008, [http://fds-smv.googlecode.com/svn/trunk/FDS/trunk/Manuals/All\\_PDF\\_Files/FDS\\_5\\_User\\_Guide.pdf](http://fds-smv.googlecode.com/svn/trunk/FDS/trunk/Manuals/All_PDF_Files/FDS_5_User_Guide.pdf).
- Mowrer, Frederick W., Ph.D. P.E. and David W. Stroup P.E. *Features, Limitations, & Uncertainties in Enclosure Fire Hazard Analysis*. Gaithersburg, MD: National Institute of Standards and Technology, 1998, <http://www.fire.nist.gov/bfrlpubs/fire98/PDF/f98070.pdf>.

Olenick, Stephen M. and Douglas J. Carpenter. "An Updated International Survey of Computer Models for Fire and Smoke." *Journal of Fire Protection Engineering* 13, (May, 2003): 87, [http://www.firemodelsurvey.com/pdf/Olenick\\_Carpenter\\_JFPE\\_87-110.pdf](http://www.firemodelsurvey.com/pdf/Olenick_Carpenter_JFPE_87-110.pdf).

Walton, William D. and Philip H. Thomas. "Estimating Temperatures in Compartment Fires." Section 3, In *The SFPE Handbook of Fire Protection Engineering*. 3rd ed., 175-176. Quincy, MA: National Fire Protection Association, 2002.

## Appendix A: Grid refinement results

Given in this appendix are all of the temperature contour plots from the grid refinement study. As mentioned in section 4.1 of this report, simulations were run with five different mesh sizes ranging from 0.25m to 0.04m. The contour plots shown in the section were used to compare the results of each simulation and determine the appropriate mesh size for this project. It is clear from Figure 32 that the 0.25m grid was too coarse to get a good indication of what was happening in the model. Figures 33-36 illustrate that the results become more accurate as the mesh size is reduced. Once the mesh was reduced to smaller than 0.05m, the trade-off between computation time and accuracy became too great.

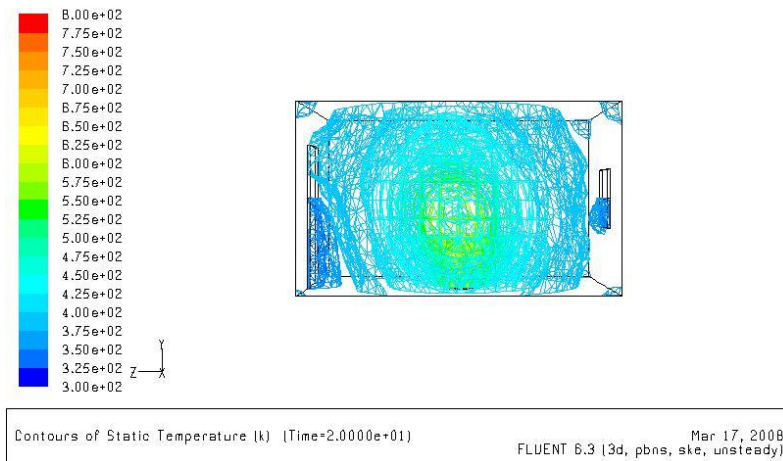


Figure 33. Temperature contours for 0.25m grid

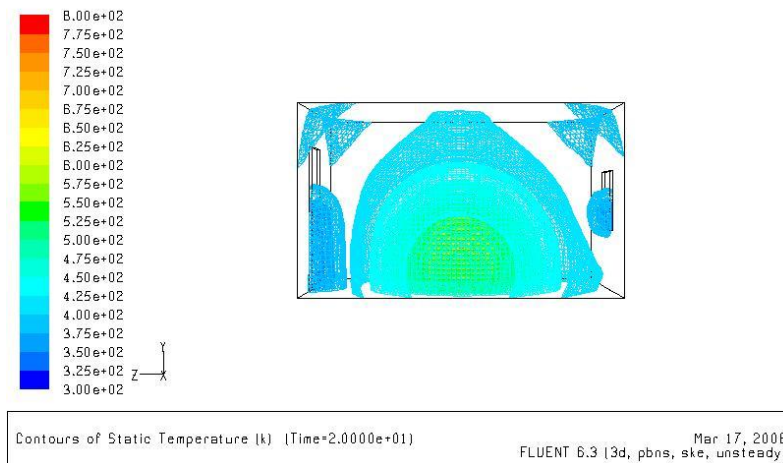


Figure 32. Temperature contours for 0.10m grid

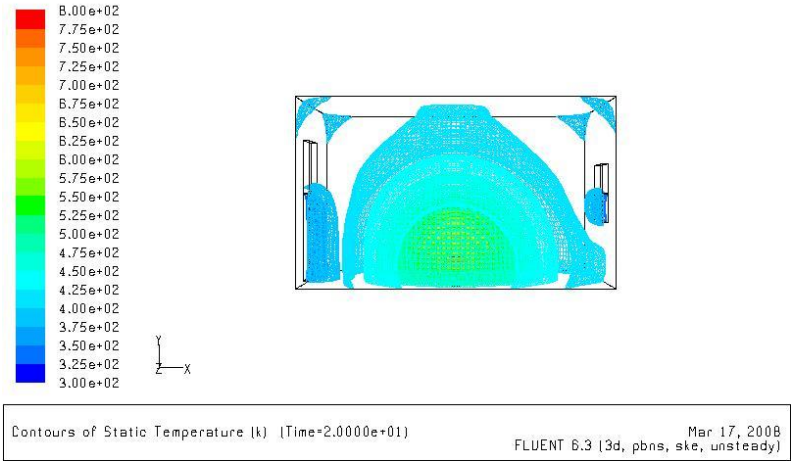


Figure 36. Temperature contours for 0.075m grid

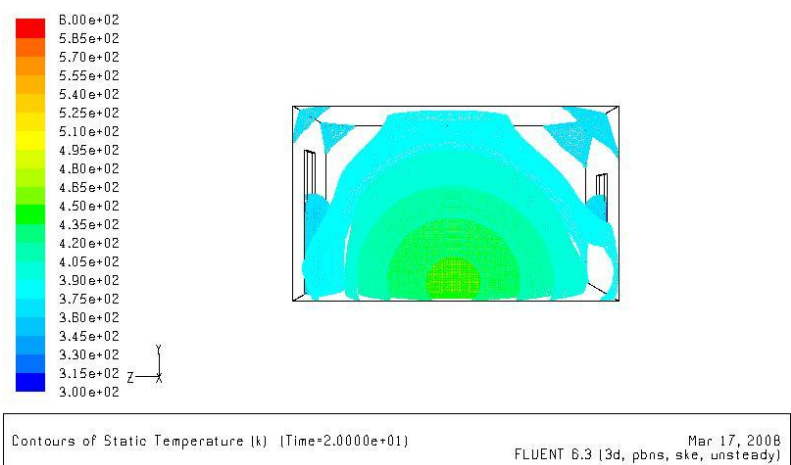


Figure 35. Temperature contours for 0.05m grid

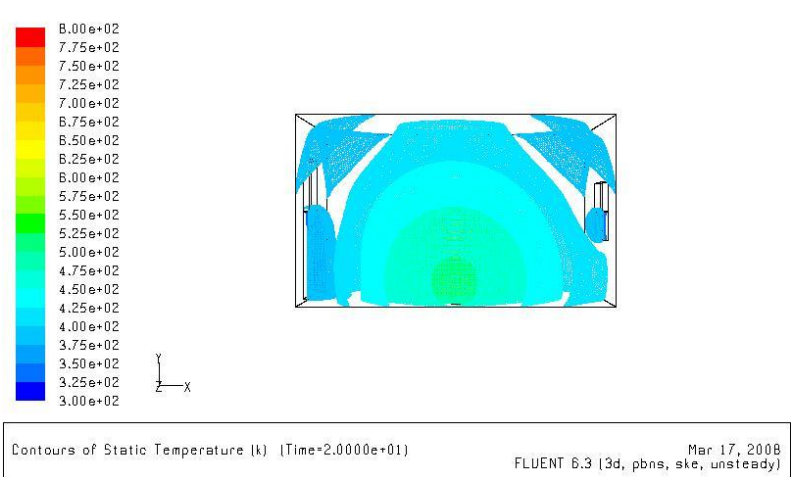


Figure 34. Temperature contours for 0.04m grid