Smoke reservoirs

-an evaluation of CFD-modeling as a design tool

Jakob Hagman
Fredrik Magnusson

Department of Fire Safety Engineering
Lund University, Sweden

Brandteknik
Lunds tekniska högskola
Lunds universitet

Report 5130, Lund 2004

Smoke reservoirs
-an evaluation of CFD-modelling as a design tool

Jakob Hagman
Fredrik Magnusson

Stockholm/Lund 2004

Smoke reservoirs – an evaluation of CFD-modelling as a design tool

Jakob Hagman
Fredrik Magnusson

Report 5130
ISSN: 1402-3504
ISRN: LUTVDG/TVBB--5130--SE

Number of pages: 74

Keywords
CFD, Computational Fluid Dynamics, FDS, Fire Dynamics Simulator, Smoke reservoirs, Stratification.

Sökord
Brandgasberäkningar, Atrium, Köpcentrum, Brandgassektionering.

Abstract
This report evaluates the methods used to predict smoke movement in atriums and the limits of smoke reservoirs used today. CFD-modeling is a wide spread tool for these types of calculations. The results of this project shows that the most important issue is to know what the user put in to the model. Most important of all is to know if there is a temperature gradient over the height of the atrium and if so what this gradient looks like.

Brandteknik
Lunds tekniska högskola
Lunds universitet
Box 118
221 00 Lund

brand@brand.lth.se
http://www.brand.lth.se

Telefon: 046 - 222 73 60
Telefax: 046 - 222 46 12

Department of Fire Safety Engineering
Lund University
P.O. Box 118
SE-221 00 Lund
Sweden

brand@brand.lth.se
http://www.brand.lth.se/english

Telephone: +46 46 222 73 60
Fax: +46 46 222 46 12

Summary

This report contains the work conducted within the course Problem based fire and risk management, VBR 131, at the Department of Fire and Safety Engineering at Lund University. The course marks the end of the Fire and Safety Engineering program and aims to show the students abilities in working independently and apply the knowledge acquired during the years at the University. The project has been conducted with the help and support of the Department of Fire and Safety Engineering and WSP Fire.

The aim has been to evaluate the arbitrary limits regarding the size and dimensions of smoke reservoirs in atriums used in the UK. These are similar to the ones used in other countries in fire safety design. The guidelines recommend that the smoke reservoirs should have a maximum area of 1000 m^2 alternatively the length should not exceed 60 meters when natural smoke ventilation is used. These limits are set to ensure that the smoke from a fire is not cooled to an extent whereas the smoke loses too much thermal buoyancy and thereby becomes difficult to ventilate and risk spreading to other parts of the building. Whether the size of the reservoirs is the most important factor when it comes to cooling of the smoke or if other factors make a larger contribution was also evaluated. To allow a more flexible design of atria a less limited method of deciding the sizes of smoke reservoirs would be of great interest for fire safety engineers.

To evaluate the current rules, simulations of smoke movement in atria haven been performed with the Computational Fluid Dynamics (CFD) code Fire Dynamics Simulator (FDS). Simulations were performed where the impact of the size of the reservoirs for the two different geometries were evaluated. Further more the effect on the smoke from temperature differences withing the atria as well as exposure to cold surfaces.

The results of the simulations shows, under the given circumstances of the conducted simulations, that using these limits does ensure that the smoke is kept at the top of the atrium, which allows an effective smoke ventilation and a minimum of smoke spread. However the results also suggest that the recommended maximum area and length should not be considred to be definite upper limits. Instead other factors, such as the initial temperature within the atrium space, was shown to have a greater impact on the behaviour of the smoke within the atrium than the physical size of the reservoir.

As the work evolved it soon became clear that original goal of coming up with a general method or formula to decide the size of smoke reservoirs would not be possible within the limitations of this project. However the project resulted in a preliminary study and evaluation of the current rules an could serve as a basis for further works in this field. During the studies a number of variables have been identified that have a great impact on whether or not an alternative design of can be allowed. If an alternative design of the smoke reservoirs within an atrium is suggested this should always be verified with a CFD-simulation of a fire scenario for the specific case. To simplify this process this project has resulted in a number of points to consider when using CFD as a design tool, these are:

- Always include the specific meteorological conditions at the location in your model
- Define a fine grid for areas close to the fire and ensure that the solution is grid independent
- If possible use ventilation data on the internal climate since initial temperature differences can have a more extensive effect on the behaviour of smoke than the physical size of the reservoir.
- Increase the size with caution since the effectiveness of smoke vents decrease as the smoke is cooled.
- Since there are always a certain amount of uncertainties involved in fire modelling a margin of error must always be included in the calculations and the conclusions from the results.

It is important to remember that these recomendations are the results of the modelling done within this project and under the given circumstances therein. It has not been possible to analyze the different parameters to the extant necessary to completely overlook them in the design of these smokereservoirs.

Sammanfattning

Följande rapport redovisar det arbete som utförts som en del i kursen Brandtekniskt Projektarbete, VBR 131. Kursen skall ses som en avslutande del på Brandingenjörsprogrammet där studenterna på ett självständigt sätt skall tillämpa de kunskaper som förvärvats under utbildningen. Projektet har genomförts i samarbete med WSP Brandteknik.

Syftet med projektet var att utvärdera de regler angående storlekar på brandgasreservoarer som används i hög utsträckning, främst i Storbritannien, när brandgashantering projekteras i atrier. Dessa regler föreskriver att reservoarerna maximalt får vara 1000 m^2 alternativt 60 meter långa då naturlig brandgasventilation används. Detta för att brandgaserna inte skall kylas i alltför hög utsträckning så att de förlorar sin termiska lyftkraft. Om så är fallet kan en effektiv ventilation av brandgaserna omöjliggöras och dessutom riskeras spridning av brandgaser till angränsande delar av byggnaden. Huruvida storleken på reservoarerna är den viktigaste faktorn i fråga om kylning av brandgaserna eller om andra faktorer påverkar i högre utsträckning var en del av frågeställningen. För att möjliggöra en större flexibilitet i designen av framförallt nya köpcentrum skulle en mindre begränsad metod för att bestämma storleken på reservoarerna vara av stort intresse för brandskyddsprojektörer.

För att utvärdera de gällande reglerna har simuleringar av brandgasspridning i atrier genomförts med hjälp av Computational Fluid Dynamics (CFD) programmet Fire Dynamics Simulator (FDS). Simuleringar genomfördes där storleken på reservoarerna för de två geometrierna varierades. Dessutom utvärderades inverkan av temperaturskillnader i rummet och avkylning mot kalla ytor på brandgasernas beteende med hjälp av FDS.

Resultaten från CFD-simuleringarna visar, under de förutsättningar som gällde vid simuleringarna, att om dessa gränser används bildas ett brandgaslager i toppen av atriet, vilket tillåter en effektiv brandgasventilation och en minimal brandgasspridning. Vidare pekar resultaten på att de gränser på 1000 m^2 area och 60 meters längd inte är att betraktas som några absoluta övre gränser. Istället visade resultaten att andra faktorer, såsom temperaturskillnader i rummet skall beaktas då de i många fall har större betydelse för brandgasernas beteende än reservoarernas storlek.

Allt eftersom arbetet fortskred stod det snart klart att den ursprungliga målsättningen att framställa en allmängiltig metod för att bestämma storleken på reservoarerna inte skulle vara möjligt inom ramarna för detta projekt. Däremot resulterade arbetet i en inledande studie och utvärdering/granskning av de befintliga reglerna som kan ligga till grund till fortsatta studier inom detta område. Under studierna har ett antal variabler identifierats som har stor betydelse för huruvida alternativa utformningar kan tillåtas. I sådana fall bör utformningen verifieras med CFD-modellering av den specifika lokalen. För att förenkla denna process har de erfarenheter som dessa studier givit lett fram till ett antal punkter att tänka på då CFD används som ett hjälpmedel vid design av brandgasreservoarer, dessa är:

- Ta alltid hänsyn till de lokala meteorologiska förhållandena i modellen.
- Använd mindre kontrollvolymer för områden nära branden och kontrollera att lösningen är oberoende av modellens upplösning.
- Undersök inomhusklimatet i det aktuella projektet. Det har visat sig att en stor temperaturskillnad mellan botten och toppen av atriet har större effekt på brandgasernas beteende än brandgassektionens storlek. Information som denna finns ofta hos ventilationsprojektören.
- Vidta alltid försiktighet vid förstoring av brandgassektionerna då brandgasventilationens effektivitet i mångt och mycket är beroende av brandgasernas lyftkraft.
- Det bör alltid finnas en säkerhetsmarginal i beräkningar av den här typen då det inte kan förutsättas att modellerna är helt exakta.

Viktigt att nämna är att de rekommendationer som ges utifrån den modellering som utförts baserar sig på de givna förutsättningar som gällde för det här projektet. De parametrar som i projektet hade mindre betydelse för brandgasernas beteende har inte studerats i den utsträckning att man helt kan avskriva dessa vid projektering av brandgasreservoarer.

Table of contents

Acknowledgements

This thesis was conducted to meet the requirements for the Bachelor of Science degree in Fire Protection Engineering at the Department of Fire Safety Engineering at Lund University, Sweden. We would like to recognize the people, who has been involved in the project and helped us along the way.

Kevin McGrattan has been of great help for us in the process of understanding how the CFD software works and how to use it. Mr. McGrattan also provided priceless insight to the Barbers Point experiment.

Bo Svensson at ScheiwillerSvensson Architechts provided a great general view of the design of shopping malls.

Daniel Gojkovic for his patience and help throughout the project.

We would also like to send our thanks to **Tanaka Takeyoshi** at Kyoto University and professor **Wan Ki Chow** at the Hong Kong Polytechnic University for their help in the preparation phase of this project. They have provided some very useful input, especially during the validation work.

Last but not least we would like to recognize **Per Olsson** and **Simon Lay** at WSP Fire for their efforts to support us in our work and providing the tools necessary.

Jakob Hagman and Fredrik Magnusson, Stockholm August 2004

1 Introduction

This section gives an introduction to the background, purpose with the study and the methodology used in the project. Also, the overall limitations of the project are discussed.

1.1 Background

When designing large premises such as warehouses, malls, etc. smoke control can pose a problem. Certain arbitrary limits for the size of smoke reservoirs have been set to prevent the smoke, which would be produced in case of a fire, from being cooled down by the surroundings. The volume and surface areas, which the smoke will come in contact with, can become too large. If so the smoke is cooled and the buoyancy decreases, which prevents the smoke from forming a layer of hot gases at the top of the enclosure. This makes it hard to extract the smoke through the smoke management systems.

The 1000 m^2 rule for smoke reservoirs utilizing natural ventilation is well known as an arbitrary limit, this is increased to 1300 m^2 in the presence of mechanical ventilation. This has been used for many years in fire safety design of new buildings. There is a similar arbitrary limit when it comes to the smoke reservoir length. This is set at 60 m. These limits can be found in e.g. BRE 186 [1].

These limits have perhaps been questioned before, but existing tools to deal with this are limited in terms of application and accuracy. The problem has been the cost of doing the research needed in order to increase, or perhaps decrease, them. Instead of saving money on e.g. a smaller number of smoke curtains these well known limits have been used to get around the costs of the research.

This project is a part of the course Problem based fire and risk management, VBR 131, given at the Department of Fire Safety engineering at the University of Lund. The fire and safety engineer student will in this course show his ability to apply his knowledge in fire safety design as well as risk management, which has been acquired during the time at the University of Lund. The project should contain independent analyses on the subject and will result in a scientific report and a presentation. The project will be executed in close cooperation with WSP Sweden AB, Stockholm.

1.2 Purpose

The aim is to look at the different physical variables affecting smoke movement in large volumes using the tools for predicting smoke behaviour available today. This is done to be able to give a number of recommendations to engineers working on and evaluating smoke management in this sort of buildings. Answer the question whether or not the size of the smoke reservoir the most crucial variable when looking at stratification.

General purpose

- Give a number of recommendations for smoke design using CFD in general and FDS in particular.
- The recommendations should be valid in various cases.
- There should be a possibility to present the work nationally as well as internationally.
- Create competence within an area that to this date needs more research.

Questions

- Which are the different things that could the affect the smoke stratification?
- What kind of buildings should be studied?
- Could the project result in some sort of classification?
- How can the recommendations be validated?
- How can the result of the study be presented?

1.3 Methodology

The overall methodology structure used in this project is presented schematically in figure 1.1, below. The first thing to do was a thorough literature study of work done in the past. Swedish guidelines, Boverkets Byggregler [2], do not include these limits, even though limits of similar kind are being used. The authors turned to looking at British and American documents where the limits are described more in detail, especially in the BRE [1]. Digging into the source documents the search ended up back in the sixties and seventies.

There were also literature studies made on the Fire Dynamics Simulator [3, 4] and the validation of the program [5, 6, 7]. The authors found that there has been some validation work done on the particular fires used in this project, but decided to conduct a validation of their own. The validation included computer modelling, hand calculations and comparisons to real life experiments at Barbers Point, Hawaii. The results of this is described in chapter 3 of this document.

The validation work was not only a way to prove that the CFD program handled the small fires in large spaces, but it also served as a way to get used to the CFD software. There were also some great opportunities, presented by WSP Fire, to model fires, using FDS, for projects both in Sweden as well as in China and Great Britain. This work has provided enough confidence for the project in all.

Next step in the project was to identify which geometries and what cases to study. Therefore a lot of time was spent talking to fire engineers with long experience of working with shopping malls and other buildings with large volumes. Architect Bo Svensson [8] provided great input on the typical geometries of such buildings. ScheiwillerSvensson has among others designed Kista Shopping Centre, one of Sweden's largest shopping malls.

The last step of the preparation phase was to get the model running. A number of basic tests were run in FDS to mark the starting-point of the computer modelling.

Looking at the results of each computer run gave a hint on what to examine next. Different factors such as ambient temperature, surfaces temperatures and volumes were varied to understand what contributes to the stratification phenomena.

At the end the results was summarized into this report and the conclusions from the work are shown in chapter 8.

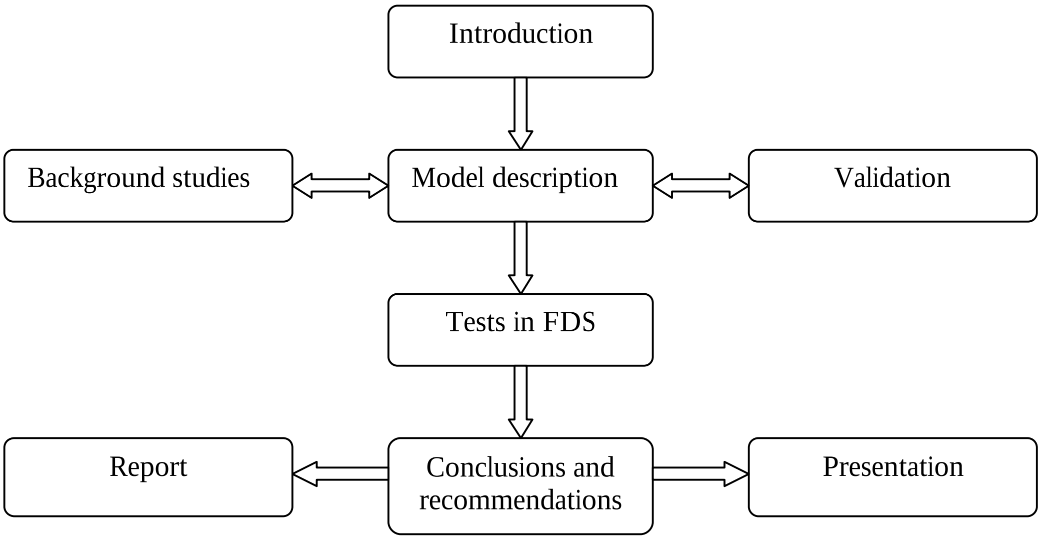


Figure 1.1, Structure of the project 'Smoke Reservoirs'

1.4 Limitations

Deciding what the limits should be for a project of this kind is probably one of the most important questions.

Fire safety engineering is not and probably never will be an exact science. This is due to the complexity of the physical phenomena that occur in a fire. How fires develops and smoke spreads depends on an almost infinite number of parameters that change over time. Fire modelling therefore depends to a large extent on assumptions and simplifications of reality. However, the art of predicting the behaviour of fires has come a long way during the last few years largely due to the development of Computational Fluid Dynamics.

One important limitation in this project is the computer software used. Even though a lot of time has been put in to the validation process one can never look at the results without a large amount of caution. To interpret them one should have good knowledge of fire behavior in general and CFD modeling in particular.

As the purpose of this study was to evaluate the possibilities to use CFD modeling as a design tool the emphasis has been on performing calculations with FDS. In the validation chapter however the results of the simulations has also been compared to the results of hand calculations performed with correlations used to predict plume characteristics as well as the experimental. The results of these comparisons have served as a background to the further simulations, i.e the same differences were expected in these cases as the cases evaluated were similar to the validation case.

Another important issue is the authors' limited experience of CFD modeling prior to working on this project. Using tools like these take quite some time to getting used to and skill is often linked to the amount of experience of the engineer. However the authors have used the methods to the best of their abilities and the work has been conducted in a way similar to the one used by fire engineers in everyday projects.

2 CFD model

This section describes the computational fluid dynamics model FDS that was used in the project. It gives a short introduction to fire modelling tools, the calculation methods used in the model, the different parts of setting up and running a model and interpreting the output. Limitations in the model are also discussed.

2.1 CFD

To be able to predict the growth and spread of a fire is crucial in order to give a building the right design from a fire safety perspective. The development of faster and better computers has allowed the models to calculate smoke transport, and other phenomena associated with fire, to become more complex. The latest tool available to fire consultants is a type of model called Computational Fluid Dynamics also known as CFD. CFD has been around for a few years already, but has just recently started to become more accessible to fire engineers.

There are a number of CFD field models on the market today. Computer programs such as CFX [9], Sofie [10] and FDS [11] are already well known tools. In a few years time it is likely that the use of CFD software will be similar to the use of the two-zone models used today within fire engineering.

2.2 Fire Dynamics Simulator Version 3.1

The primary tools that will be used in this project are Fire Dynamics Simulator and the visualization program Smokeview [11]. These programs have been developed by the National Institute of Standards and Technology and are so called freeware, which means that they are available to anyone. This vouches for a wide use of the program and secures a continuous development of the software.

FDS is a computational fluid dynamics model of fire driven flow. The program numerically solves a form of the Navier-Stokes equations appropriate for low-speed, thermally driven flow with an emphasis on smoke and heat transport from fires. FDS, as well as the other CFD software, consists of three parts, the pre-processor, the calculation software and the postprocessor. For a more thorough description of the program it is recommended to read the technical reference guide provided by NIST and Kevin McGrattan [3].

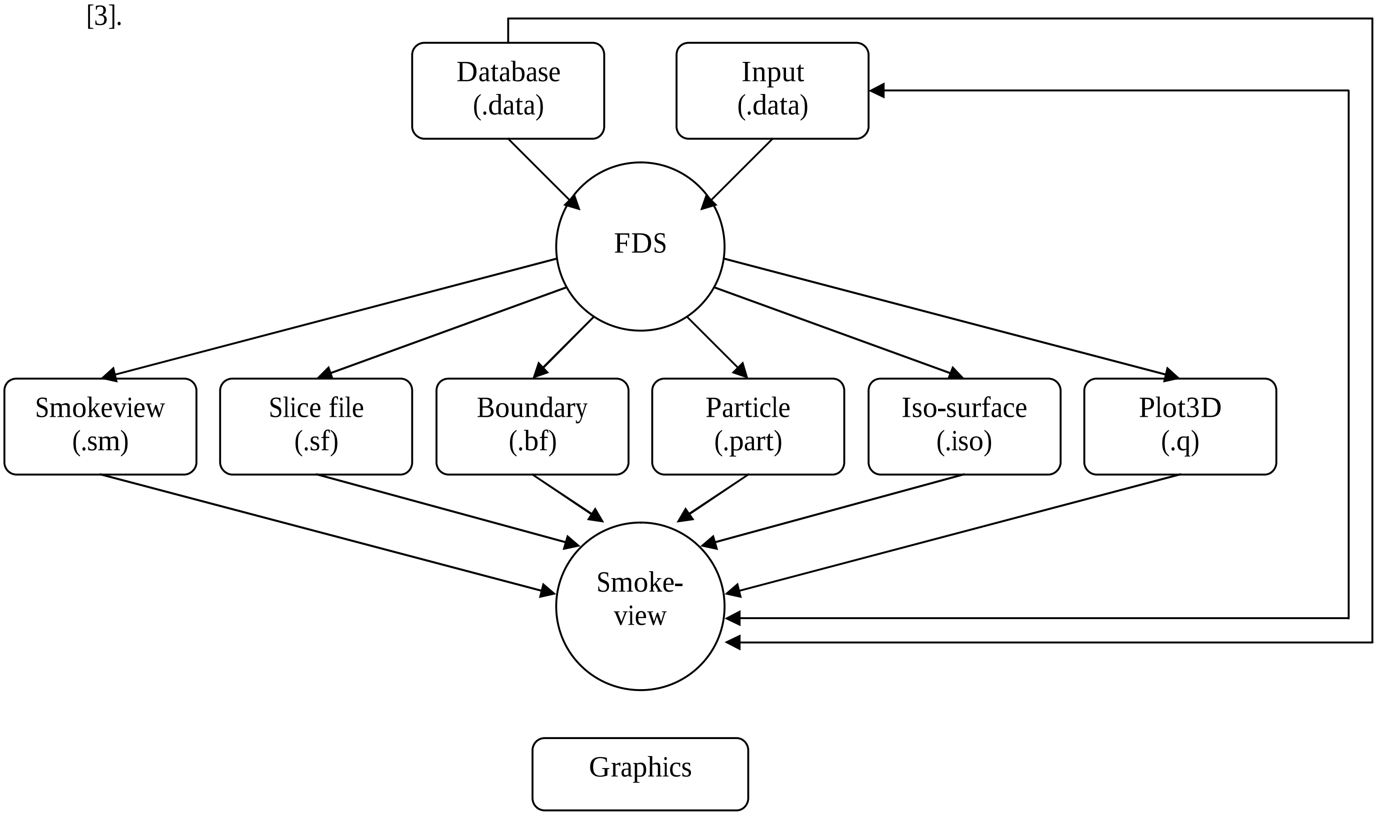


Figure 2.1 Diagram illustrating data files and programs used in the NIST Fire Dynamics System.

The model was originally designed to analyse industrial-scale fires and is considered reliable when the fire size is specified and the building is relatively large compared to the fire. In these cases the model predicts flow velocities and temperatures to an accuracy of 10 to 20 percent compared to experimental measurements [5, 6]. These conditions are similar to the cases that are studied in this project, and therefore FDS should be an ideal tool to use for this purpose. As FDS was originally designed primarily as a tool to predict the transport of heat and smoke from a fire the code has for this purpose undergone a considerable amount of validation work.

2.2.1 Equations of conservation

The CFD models are, in general, based on a number of equations for the conservation of different factors such as mass and energy. This basically mean what goes in must come out. In the CFD model used for this project there are four basic rules for the conservation [3]. For a more thorough description of the equations look to the technical manual provided by NIST [3].

Conservation of mass

$$\frac{\partial r}{\partial t}+\nabla\cdot ru=0$$

Conservation of species

$$\frac{\partial}{\partial t}(rY_l)+\nabla\cdot rY_l u=\nabla\cdot rD_l\nabla Y_l+\dot{m}_l'''$$

Conservation of momentum

$$r\left(\frac{\partial u}{\partial t}+(u\cdot\nabla)u\right)+\nabla p=rg+f+\nabla\cdot t$$

Conservation of energy

$$\frac{\partial}{\partial t}(rh)+\nabla\cdot rhu=\frac{Dp}{Dt}-\nabla\cdot k\nabla T+\sum_l\nabla\cdot h_l\,rD_l\nabla Y_l$$

Where:

ρ = density
t = time
u = velocity vector (u,v,w)
Y_l = the mass fraction of the l-th species
D = the diffusion coefficient
$\dot{m}_l'''$ = production of l-th species per unit volume
p = pressure
f = an external force e.g. from sprinklers
τ = viscous stress tensor
h = enthalpy
q_r = radiative heat flux
k = thermal conductivity
T = temperature

2.2.2 Combustion

The model used for this project uses two different types of calculations to predict combustion. The method chosen by software is dependant on the resolution of the grid. A fine grid enables the program to do a DNS calculation where the diffusion of fuel and oxygen are modelled directly. However, this approach is computationally very expensive, as it requires very small length and time scales to be captured by the model. Therefore it is to date not practically applicable when modelling large case fire scenarios.

In the case with coarser grid, which is the case in the simulations performed in this project, the software uses a LES calculation to predict the diffusion of fuel and oxygen. From that result a mixture fraction model can be used to predict the combustion. The mixture fraction combustion model is based on the assumption that large-scale convective and radiative transport phenomena can be simulated directly, but physical processes occurring at small length and time scales must be represented in an approximate manner. These assumptions were considered to give sufficiently accurate results for the purposes of this study.

2.2.3 Convective heat transfer to walls

The method for calculating the convective heat transfer in FDS is also dependant on if a DNS or LES is performed. If a DNS calculation is performed the convective heat flux can be calculated directly from the gas temperature gradient at the boundary:

$$\dot{q}''_c = -k \frac{\partial T}{\partial n}$$

Where n is the spatial coordinate pointing into the solid. In the LES calculations used in the simulations performed in the simulations performed in this project the heat flux is obtained from natural and forced convection correlations:

$$\dot{q}''_c = h \Delta T \quad \mathrm{W/m^2}$$

Where ΔT is the difference between the wall and the gas temperature and h is calculated from the following equations for natural and forced convection respectively

$$h = C\left|\Delta T\right|^{1/3} \qquad \text{or} \qquad h = \frac{k}{L} 0.037 \,\mathrm{Re}^{4/3}\,\mathrm{Pr}^{1/3}$$

whichever is the greatest. C is the coefficient for natural convection (1.43 for a horizontal surface, 0.95 for a vertical). L is the characteristic length of the obstruction, k is thermal conductivity of the gas and the Re and Pr numbers are based on the gas flowing past the obstruction.

2.3 Method

Figure 2.1 shows a schematic picture of how to work with FDS. A more detailed description of every step in the process of doing calculations using FDS is given below. For further information look to the FDS user's guide [4].

2.3.1 Pre-processor

The pre-processor in FDS is where you define grid, geometries, fire size and growth rate, surface material and type of fuel. This is also where you define the different output data desired for the particular run. The input data is put together to an data-file, which is basically a text file consisting of command lines for the different functions and parameters in FDS, that will generate all calculations specific for the simulation. An example of a data-file and as well as a brief explanation of each of the defined parameters is presented in Appendix A.

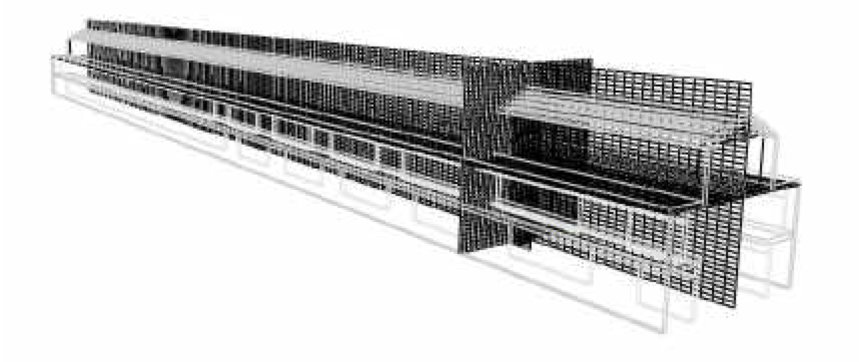


Figure 2.2, An illustration of the grid

2.3.2 Calculation software

FDS itself is what one might call the calculation software. This is where the data-file is processed and a number of partial differential equations are solved. Together with several equations to calculate turbulence, radiation, soot production, etcetera, the output-file is formed. This is all done using a high number of iterations, which is why the CFD program is relying on a computer that can handle this huge amount of calculations. In some large and complex geometries the calculations can be running for weeks even when using the latest processors available.

In this project a single 2.4 GHz Pentium4® processor with 1GB RAM was used for calculations. In some of the larger calculations performed to investigate whether the solutions were grid independent the simulation times approached one week.

The use of parallel processors is currently under development at NIST and this would mean that the computer time required for calculations could be significantly shorter in the near future.

2.3.3 Post-processor

Smokeview is the post-processor, which is used together with FDS. This program allows the user to look at the different phenomena linked with fires in enclosures. Visibility, temperature, flow, radiation and CO-levels are shown both in 2 and 3-D as a function of time.

2.4 Out data

When looking at the data provided by FDS in Smokeview there are a number of parameters that are of interest. From the list of about twenty parameters ranging from wall temperature to fractions of O_2 this work is limited to looking at heat release, temperature, visibility and particle movement in the enclosure. These are presented in different ways.

Particle files – The files contain the locations of tracer particles used to visualize the flow field. Particles may be coloured with various gas properties. Sprinkler water droplets if present are coloured blue. This type of file is an excellent tool to look at the movement of smoke throughout the enclosure.

Slice files – Slice files contain data recorded within a rectangular array of grid points at each recorded time step. Continuously shaded contours are drawn for simulation quantities such as temperature, gas velocity and heat release rate. All or part of a plane is selected when setting up the FDS input data file. In this project the slice files are used to illustrate temperature and visibility in particular.

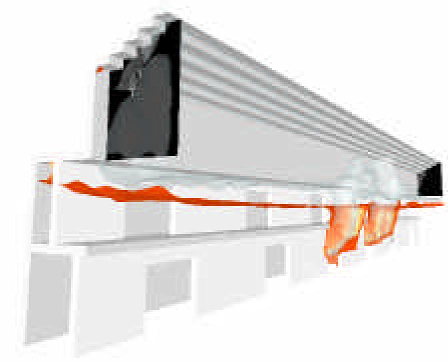


Figure 2.4, FDS Isosurface

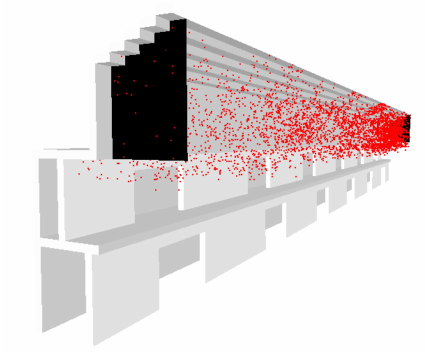


Figure 2.3, FDS Particle file

Isosurface – The surface where a quantity such as temperature attains a given value is called an isosurface. An isosurface is also called a level surface or 3D contour. Isosurface files contain data specifying isosurface locations for a given quantity at one or more levels. These surfaces are represented as triangles.

Plot3D – Data stored in Plot3D files use a format developed by NASA and are used by many CFD programs for representing simulation results. Plot3D files store five data values at each grid cell. FDS uses Plot3D files to store temperature, three components of velocity and heat release rate. Other quantities may be stored if desired. An FDS simulation will typically create Plot3D files at several specified times throughout the simulation.

2.5 Validation

To have confidence in using FDS for a particular geometry with a certain fire size it is important to validate the model against a real experiment. Even with a well-executed validation the output should always be looked upon with great caution since the three-dimensional output can look very realistic to the naked eye. For this project the validation was done using an experiment executed in an old aircraft hangar. The validation work is described in chapter 3.

2.6 Limitations in FDS

The first problem a new user of FDS encounters is the tedious process of putting together the indata-file. This contains of a text-file with different command lines for different functions in FDS. With the help of excel spreadsheets and as the user gains experience the time spent compiling input data is shortened.

There are also limitations within the software as assumptions and simplifications of reality are inevitably made. Some of these assumptions are described above in the description of the combustion and heat transfer models. Similar assumptions are made in other areas of the model as well. Due to limitations in the model algorithms or numerical grid the parameters of interest may not be solved. Also there is the possibility for numerical errors as the algorithms are solved. The user should always be aware of these limitations in the program.

Furthermore the program is always dependant on reasonable assumptions and the output should always be considered with a certain amount of criticism, if something doesn't look right one should be open to reviewing the input.

‘The equations and numerical algorithm described in this document form the core of an evolving fire model. As research into specific fire-related phenomena continues, the relevant parts of the model can be improved. Because the model was originally designed to analyze industrial scale fires, it can be used reliably when the fire size is specified and the building is relatively large in relation to the fire. In these cases, the model predicts flow velocities and temperatures to within 20 % of the experimental measurements. In cases where the fire is large relative to the enclosure, the uncertainty of the model is greater due both to the lack of input data for material properties and combustion chemistry and to greater numerical error in combustion and radiation transport.’

‘Any user of the numerical model must be aware of the assumptions and approximations being employed. There are two issues for any potential user to consider before embarking on calculations. First, for both real and simulated fires, the growth of the fire is very sensitive to the thermal properties of the surrounding materials. Second, even if all the material properties are known, the physical phenomena of interest may not be simulated due to limitations in the model algorithms or numerical grid. Except for those few materials that have been studied to date at NIST, the user must supply the thermal properties of the materials, and then validate the performance of the model with experiments to ensure that the model has the necessary physics included. Only then can the model be expected to predict the outcome of fire scenarios that are similar to those that have actually been tested.’

K. B. McGrattan, et al.

3 Validation

When using a tool like the CFD program FDS it is crucial that a proper validation has been conducted and presented. Validation is a process that is used to ensure that a model gives correct results for the phenomena that it is developed to predict. This usually consists of simulating an experiment and comparing the results from the model with the data recorded in the experiment. In the case of validating a fire model a big issue has always been to provide good experimental data, the complexity of fire and the large number of variables that affects the development of the fire makes it hard to perform repeatable experiments. The shape and form of the validation can vary quite a lot, but there is one important thing one should strive for. That is to use a validation that represents a similar scenario to that one intends to look at.

To validate the CFD model used in this project a large-scale experiment was used. This was done comparing the results from the actual test with the output from the computer model. The idea was to investigate if the model tend to over- or underestimate any of the parameters. To fit the purposes of this project an experiment with a relatively small fire in a large volume was picked. The choice of experiment fell on one done in a Navy hangar at Barbers Point, Hawaii [5].

As an extra precaution some additional validations have been studied to ensure the applicability of the computer program to this particular situation.

3.1 Experiment at Barbers Point

The hangar had a maximum ceiling height of 15.1 m with floor dimensions of 97.8 by 73.8 m. The fires used in these experiments were placed at ground level 12.2 m to the west of the centreline as shown in figure 3.1. The hangar had a pitching roof, with a slope of 3 degrees up towards the centreline in the east-west direction. The roof and the walls can be considered to be thermally thin.

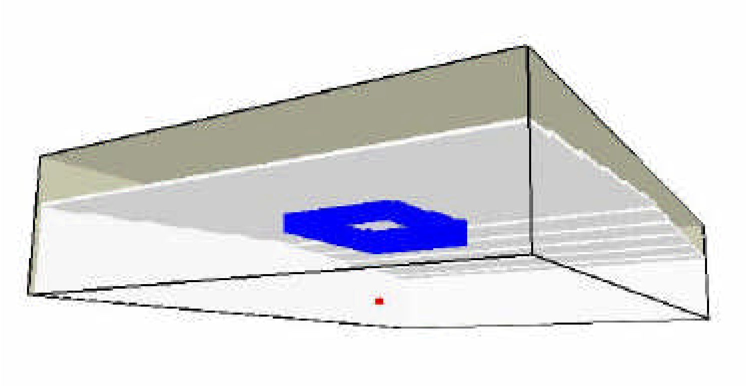


Figure 3.1, View of Hangar

Above the experimental fires there was a draft curtain mounted. The curtain and roof was sealed so that little or no smoke at all could escape through there. The dimensions of the curtain were 24.4 m in length, 18.3 m in with and 3.7 m in depth.

A number of thermocouples and flow meters were positioned throughout the hangar. The majority of thermocouples and meters were placed on the inside of the draft curtain, but also some on the outside. These were then used to monitor temperatures in the fire plume and ceiling jet as well as the speed of the jet.

There were eleven different tests conducted using JP-5. Out of these eleven tests there were two that returned reasonably reliable information on heat release rate and temperature. These two tests are the ones that have been compared to the computer model in this study. The first of the two tests used a pan with a maximum heat release rate of 500 kW. The second had a maximum heat release rate of about 2.7 MW. They will from now on be referred to as the 500 kW and the 2.7 MW tests.

The heat release rate from the two tests increased as described in table 3.1 over time and stabilized at 500 kW and 2.7 MW respectively.

Time [s]	$HRR_{500\ kW}$ [kW]	$HRR_{2.7\ MW}$ [kW]
0	0	0
1	99	803
10	not used	965
20	174	1135
50	273	1582
100	388	2139
200	481	2693
300	478	2766

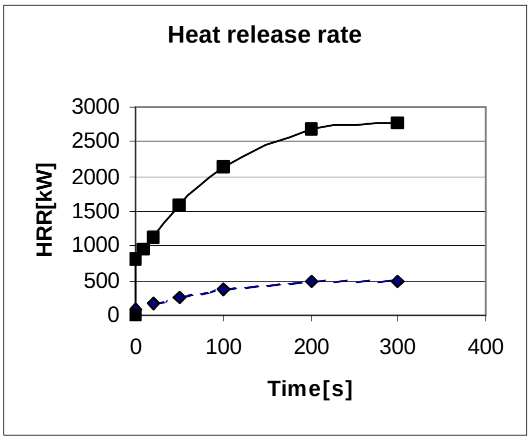


Table 3.1, Heat release rate over time [5]

3.2 Model inputs

The model inputs used to simulate the 500 kW and the 2.7 MW fires were carefully chosen so that the model would match the real life experiment as close as possible. Despite of those efforts it is crucial that one realize that we can never duplicate an experiment in a CFD model. We can only try to get as close to the conditions during the experiment as possible and then use the results with caution.

3.2.1 Geometry

The outer walls of the hangar defined the geometry, even though the experiment itself was conducted in a relatively small area of the building. The FDS grid was transformed so that the majority of cells were positioned close to the experiment, whereas the surrounding was made out of larger and fever cells. The tilting roof was defined using a stair shaped formation of blocks. With the help of the s*awtooth = .false.* command the impact of the stair formation was minimized. The draft curtain was defined by thermally thin surfaces forming a rectangle centred over the location of the experimental fires.

3.2.2 Fire

The two fires were placed as shown in the plan view of the hangar and are represented by a block. Since the surface area of the block equals 1 m^2 the heat release per area unit is the same as the desired maximum heat release. Initially the growth rate of the fire is controlled by the values given in table 6.1.

3.3 Acceptance criteria

The purpose of using a computational fire model is to make predictions on how a fire will affect a building and the people in it. In order to ensure that these predictions are valid to a satisfactory extent some sort of acceptance criteria need to be set. To this date no such general acceptance criteria has been defined which leads to difficulties to assess the performance of the FDS code.

During a fire test series in a HDR facility [7], computer models were used to simulate some of the experiments. In order to evaluate the performance of different codes a set of guidelines was developed a panel of nuclear safety and fire protection experts. The guidelines were developed with the following three issues in mind:

- Maintaining structural integrity of safety-related structures and components
- Maintaining functionality of safety-related electrical and mechanical facilities
- Maintain usability of ventilation systems and keep emergency/intervention routes smoke free (maintain their availability for use)

In the absence of other criteria these guidelines are used to evaluate the performance of FDS. The following table presents the guidelines.

Location	Parameter	Desired Accuracy
Near-Field	Temperature of hot gases, plume, etc.	± 15%
	Temperature of hot structures	± 10-15%
	Pressure	± 20%
	Fire Properties	± 30-50%
	Layer height	± 30-50%
	Event timing	± 5 minutes
Far-Field	Temperature of gases	± 20% or ± 20 °C
	Temperature of structures	± 10-15%
	Pressure difference	± 20%
	Gas concentrations	± 15-20%
	Aerosols	± 20%
	Energy and mass flows	± 30-50%
	Velocities	± 30-50%
	Layer heights	± 30-50%
	Event timing	± 5 minutes

Table 3.2, Acceptance criteria

The guideline divides the fire simulation into two different regions: the near-field and the far-field. The fire compartment and adjoining compartments where high temperatures and heat fluxes occur are considered to be near-field. In this region the temperatures and heat fluxes must be known to evaluate the effect on structures and equipment. Far-field is the rest of the building. In the far-field temperatures are not likely to reach very high levels. In stead the concern here is human survivability determined by the concentration of toxic gases. It is important that a fire computer model makes sufficiently accurate predictions of the conditions in both regions.

This guideline is used to discuss the validation work and other simulations in this project. Even though it does not necessarily represent the needed performance-based criteria, in the absence of such criteria it is used as a basis for assessing the results.

3.4 Results

The results from modelling the experiment were fairly accurate. Some variations seem to be the result of FDS over- or underestimating the value of a certain parameter. In some cases the difference between the experiment and the CFD model was due to problems to place the thermocouples in the exact position. These variations had often little or no impact on the result but in a few isolated cases they turned out to have a somewhat larger effect.

3.4.1 Plume centerline temperatures

At each evaluated time step in the 500 kW simulation FDS tended to underestimate the excessive temperature of the plume with approximately 2-3 °C. In the case of the 2,7 MW fire FDS tended to overestimate the temperatures slightly. In both cases the results were well within the margin of error compared to data from the experiments. The plume centreline temperatures were also calculated with Heskestad’s and McCaffrey’s correlations [12]. These correlations are widespread in the fire safety community and often used for hand calculations of fire behaviour. Both of them overestimated the temperatures too quite a large extent and for the cases studied results from FDS were significantly better.

Time [s]	HRR [kW]	Exp [°C]	FDS [°C]	Heskestad	McCaffrey
75	350	39 ± 2	36,3	53.9	54.7
	1775	65 ± 2	69,7	103.8	105.6
150	455	40 ± 2	37,9	59.4	59.8
	2385	71 ± 2	71,2	123.8	122.7
225	500	42 ± 2	38,6	61.6	61.9
	2700	71 ± 2	77,5	133.8	130.9

Table 3.3, Plume centerline temperatures

3.4.2 Radial temperatures

Time [s]	Radius [m]	Exp. N-S [°C]	Exp. E-W [°C]	FDS, N-S [°C]	FDS, E-W [°C]
75	1.5	38 ± 2	37 ± 2	40.7	43.8
	3.0	37 ± 2	35 ± 2	36.5	38.1
	6.1	36 ± 2	32 ± 2	35.5	35.1
	8.5	33 ± 2	31 ± 2	35.3	34.1
	9.1	not available	31 ± 2	not available	33.7
	11.6	not available	30 ± 2	not available	31.8

Table.3.4, Radial temperatures for the 500 kW fire compared with FDS

Time [s]	Radius [m]	Exp. N-S [°C]	Exp. E-W [°C]	FDS, N-S [°C]	FDS, E-W [°C]
75	1.5	62 ± 2	64 ± 2	66,3	74,9
	3.0	56 ± 2	56 ± 3	56,0	59,8
	6.1	44 ± 2	44 ± 4	53,3	53,4
	8.5	44 ± 2	40 ± 2	52,7	51,5
	9.1	not available	38 ± 2	not available	50,9
	11.6	not available	30 ± 2	not available	45,6

Table 3.5, Radial temperatures for the 2.7 MW fire compared with FDS

3.4.3 Ceiling jet

In the 500 kW case the speed of the ceiling jet is varying between 0.8 and 1.0 m/s. These numbers should be compared with the experimental results that lay between 0.5 and 0.7 m/s.

The larger fire overestimates the value of the ceiling jet velocity by approximately 0.5 m/s during the first minutes of simulation. The error increases with time.

3.4.4 Spilling and filling

Filling time was indicated by a thermocouple at the bottom of the curtain. When the temperature had increased by 0.5 °C the curtain was considered full. This definition could be questioned since it is very sensitive to the placement of the thermocouple.

The 500 kW case indicated the temperature increase at 55 seconds. Compared with the results from the experiment, which was 100 seconds, this is a fairly large difference. Looking at the simulation in Smokeview there is actually a relatively wide time span, which could be considered to be the filling time. The difference in results could be traced to the problems with visual versus measured observations. The same goes for the 2.7 MW fire.

Figure 3.2, Smoke filling with the 2.7 MW fire

3.5 Discussion

The results from the validation work cannot be considered to be a complete success. The errors increase with time and are in some cases too large to be acceptable. Despite the errors we cannot dismiss the model. The fact is that a lot of the results show that FDS handles the relatively small fire in the large compartment well. Kevin McGrattan at NIST has also expressed some scepticism when it comes to the Barbers Point experiments. According to him the conditions were not good enough to give accurate measurements.

Looking at validation work done in the past they point out that parameters such as temperature and velocity are very well predicted in conditions similar to the ones at the hangar [5].

The result of the validation done for this report, which for the parameters most important for this study provide results that meet the acceptance criteria set out (and give significantly better results than the hand calculations used), overall was positive although some of them can be questioned. Also taking into account previous validation work performed by NIST [5, 6] and that FDS was originally developed for cases similar to the ones to be studied confidence is provided to use FDS for the work done in this report.

4 Initial CFD simulations

The most important work that is done in computer modeling is correct interpretation of the results. When examining the results of the simulations that have been made the first question is. *What criteria should one look at regarding the possibility to vary the size of smoke reservoirs?* In this project the focus has been on thermal buoyancy and the stratification of the smoke. Subsequently the following question is: *How small can the temperature difference between the hot smoke layer and the cool layer be in order to ensure a two-layer configuration of the fire environment?* Results from experiments [13] suggest that even differences in the order of 10°C was enough to generate a two-layer configuration if the stability of the smoke is not upset by other factors such as outside wind pressure. The aim of this study is to investigate under what conditions, with an emphasis on the size of the reservoirs, the lack of thermal buoyancy leads to a well-mixed case, i.e. the smoke distributes evenly throughout the atrium space.

To interpret the results of the simulations the following output was studied:

- Slice files of temperature within the computational domain visualizing the temperature gradient
- Slice files of visibility within the computational domain visualizing the smoke movement
- Particulate tracings visualizing the movement of e.g. soot particles from the fire source

4.1 General

These simulations should be considered a starting point to the analysis of whether or not the size is a critical factor when designing smoke reservoirs. In these the atria were designed according to the arbitrary limits used today. In FDS the settings were left at default regarding soot production, heat release, boundary conditions etc., for further description of the default physical conditions in FDS see section 4.1.4. A relatively coarse mesh configuration is used for this study, which imposes certain limitations on the accuracy of the results. However, this preliminary simulation provides an indicative idea for further refinements to the model when a finer mesh configuration offers greater accuracy.

Primarily a scenario with a fire in a shop alongside the atrium creating a spill plume was studied, as this was considered a worst case, however tests were also performed with fires placed in the atrium creating an unobstructed plume to confirm that this assumption was valid.

The results were examined and evaluated and has been used as a comparison for the further studies with an increased area of the smoke reservoirs.

4.1.1 Geometry

The idea was to make the result from this study as general as possible, but at the same time keep it detailed so that it can be used in real projects. The focus has been on two different geometries with various measurements and proportions. The first one is the long atria, which will serve to evaluate the 60 meter-limit. The second one is the square shaped area, which for example could serve as a hub and connect these long atriums. The purpose of that would be to look at the 1000 and 1300 m^2-limit.

Before designing the two different geometries architects with special knowledge on shopping malls were consulted to get a picture of the thoughts behind the buildings [8]. The information given was then used as guide lines in the work to create two general modells to represent a typical shopping venue. It is of course difficult to create a modell that can be looked upon as general but consulting with a recognized architect gave the modell a more solid ground.

4.1.2 Design fire

The results of the simulations are to a large extent dependant on what fire load is chosen. The design fires should represent a worst likely scenario. As this study focuses on the cooling of smoke produced, a smaller fire would actually give us a worse case.

However the most important in this is that the fire load is the same in all the scenarios to make it possible to make comparisons between the different simulations. These comparisons were used to ascertain what impact the size of the reservoirs have on the results and ultimately make a prediction if this is the most important factor to consider when designing smoke management systems for malls and atria.

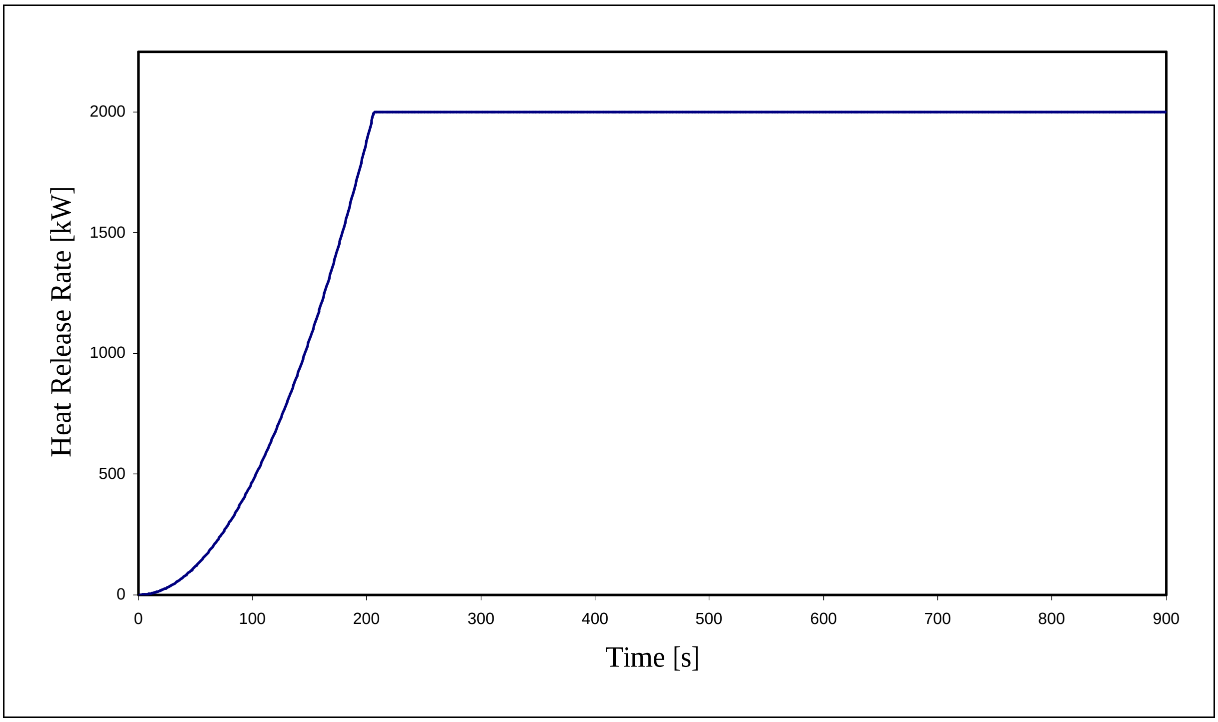


Figure 4.1, Heat Release Rate as a function of time

In this study of smoke movement within the atrium a standard design fire curve has been chosen. The fire is assumed to be a fast growing fire where the heat release rate increases with the square of time. The maximum heat release was assumed to be 2 MW. Table 4.1 describes the fire scenario.

The use of the 2MW fire is not only a conservative assumption, but also a realistic one. This since we can assume that fires in a location like the ones looked upon are most likely to be sprinkler controlled. Modern stores are very often, not to say always, sprinklered, meaning a potential fire is not likely to exceed a heat release rate higher than the 2MW used in this project.

One could argue that the sprinkler would cool the hot gases from the fire. This phenomena could very well have an effect on the results. Not having added sprinklers in the modell is an obvious limitation made to keep the workload of the project on a reasonable level. If there would be a second phase to this work the effect of sprinklers would definitly be interesting to look at.

4.1.3 Smoke management systems

The major purposes of smoke reservoirs is to contain smoke and protect areas beyond where the fire has started, if the smoke loses its buoyancy due to cooling the function of the smoke reservoir is compromised and smoke is spread to adjoining parts of the building.

A natural smoke ventilation solution was evaluated in this project. Smoke ventilation was provided with natural ventilation through smoke vents mounted in the ceiling of the atrium models. No simulations were performed with vents alongside a wall where the smoke management system would be more sensitive to an external wind pressure. The area of the vents were calculated with the iterative method described in Appendix 2 where the criteria for smoke layer height was set at 10 meters so that the smoke layer would not sink below the smoke barriers unless problems with buoyancy occurs and thereby smoke entrains the space below the smoke reservoir.

The total area of ventilation outlet was approximately 60 m^2. Inlet air was provided through doors at floor level, which were assumed to be open throughout the simulation. The approximate area of these was also 60 m^2.

4.1.4 Effect of external wind pressure

The effect an external wind pressure might have on the stability of the smoke layer was not included in this study. However experimental results suggest that even at low wind speeds, around 2 to 4 m/s, can destroy the smoke layer and lead to a smoke-logged environment. The experimental study was performed with vertical vents on the walls of the compartment, horizontal smoke vents, as in the atrium model in the ceiling are considered to be less affected by this phenomena.

4.1.5 Initial physical conditions in FDS

This section describes the default settings regarding soot production, heat release, boundary conditions etc in FDS. These settings are used the initial simulations of smoke movement in the atria, however a sensitivity analysis is to be performed to evaluate how these can affect the simulation results. If the sensitivity analysis shows that changing one or several of these parameters has a substantially negative effect this will be considered in the further studies to evaluate the size of the smoke reservoirs.

Air is the only fluid simulated within the flow domain at standard conditions. The temperature for the ambient environment has initially been specified as 20°C.

Replacement air openings have been modelled as passive openings to an infinite reservoir, i.e. no significant over/under-pressure is created with outside temperature the same as initial temperature.

4.1.6 Simulation Time

The CFD simulation of an atrium base fire was performed until “steady state” was considered to be attained.

4.2 Geometry of the long atria

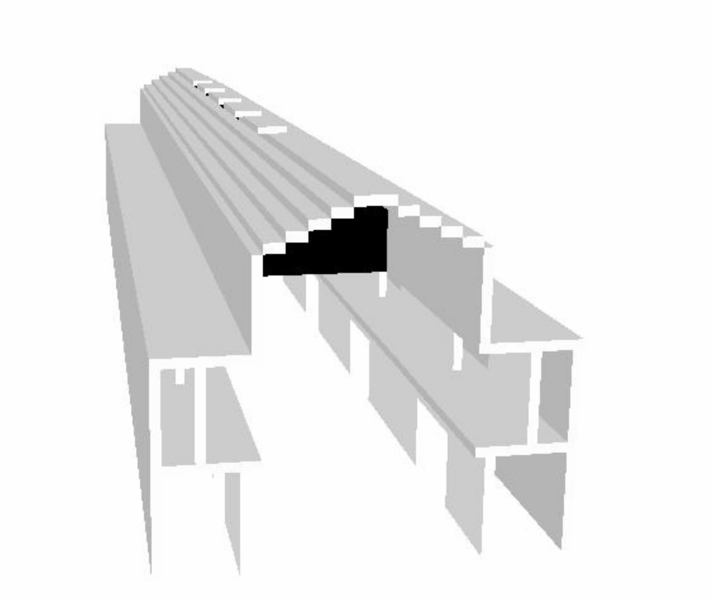


Figure 4.2, View of long atria

For the initial simulations a model of a 90 meter long, 15 meter high and 20 meters wide atrium was built. The geometry is illustrated in figure 4.2. This general atrium geometry was constructed on the basis of discussions with an architect on the design of a typical atrium [8]. It was placed in a control volume of 90 by 20 by 20 meters and the boundary to an infinite surrounding was set as open at all sides except for the bottom side of the model. The cell resolution was initially set at 0.5×0.5×0.5 meters, resulting in a model consisting of 288,000 cells.

In the first simulation smoke barriers were placed at a distance of 60 meters, the recommended maximum separation between smoke barriers in the regulatory documents [1, 14]. This was done to fine-tune the model with regards to smoke ventilation as well as to have some results to compare with the output from increased smoke reservoirs simulations

Smoke ventilation was provided via smoke vents mounted in the ceiling of the atrium model. The area of the smoke vents were initially calculated with the method of calculation described in Appendix B where the criteria for smoke layer height was set at 10 meters so that the smoke layer would not sink below the smoke barriers unless problems with buoyancy occurs and thereby smoke entrains the space below the smoke reservoir. The total area of ventilation outlet was approximately 60 m^2. Inlet air was provided through the whole cross section at each end of the atrium model, which were assumed to be open throughout the simulation.

4.3 The square atria

For the initial simulation a 35 by 35 by 25-meter cube representing a square-shaped atrium was built. The geometry is illustrated in figure x.1. It was placed in control volume of 36 by 36 by 30 meters and the boundary to an infinite surrounding was set as open at all sides except for the bottom side of the model. The cell resolution was set at 0.5×0.5×0.25 meters, resulting in a model consisting of 622,000 cells, it was soon discovered that this resolution perhaps was a bit on the optimistic side since the model took close to a week to run. However it was useful to have the results from these simulations to compare with the results with runs with a more coarse mesh configuration and thereby evaluating the effect a coarser/finer grid could have. The comparison between different cell resolutions is further described in chapter 5.5 and 7.5. The temperature was initially set at 20° Celsius both inside and outside the control volume.

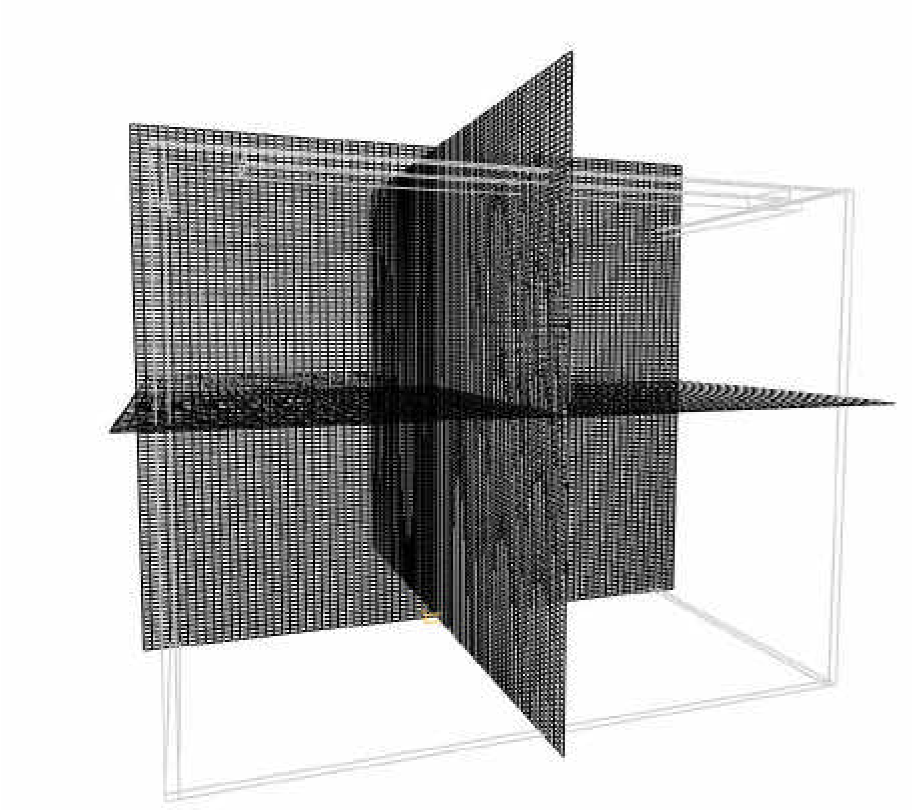


Figure 4.3, Illustration of initial geometry and grid

Smoke ventilation was provided with natural ventilation through smoke vents mounted in the ceiling of the atrium model. The area of these were calculated with the method described in Appendix X where the criteria for smoke layer height was set at 10 meters so that the smoke layer would not sink below the smoke barriers unless problems with buoyancy occurs and thereby smoke entrains the space below the smoke reservoir.

In the first simulation the area of the smoke reservoir set at 1000m^2 the recommended maximum area of smoke reservoirs in the regulatory documents [1]. This was done to fine-tune the model with regards to smoke ventilation as well as to have some results to compare with the output when increased smoke reservoirs are simulated.

5 Further CFD simulations

The scope of these studies was to evaluate what effect increased smoke reservoir size as well as changing of different physical and computational parameters might have on the results of the initial simulations. The simulations were performed in the model that was considered most sensitive to alterations of the parameter in question. This means that some of the parameters were altered in the long atrium model and other in the square model.

As a lot of parameters are uncertain there would a nearly impossible task to vary all of these in all of the scenarios considered. This project does not include all the possible alterations to the physical conditions that might occur in real life but does however investigate the ones that with a initiated eye would appear to have the most significant effect on smoke management.

5.1 Increased reservoir length

A simulation was performed in the initial atrium model as described previously with the difference that smoke screens were moved and placed at a distance of 90 meters, 1.5 times the recommended distance. This was considered a reasonable first increase of reservoir size.

The length of the initial model was then increased to 180 meters. The cell resolution was set at 1×1×0.5 meters so as not to get unreasonably long simulation times, however further simulations were also performed where the grid was refined in areas close to the fire (see chapter 4.5-Further Simulations). A distance of 180 meters (3 times the recommended distance) between smoke barriers was simulated.

5.2 Increased reservoir area

In this simulation a 5000 m^2 smoke reservoir, which is five times the recommended size, was evaluated. All other conditions were kept the same as in the initial simulations.

5.3 Simulation of temperature differences within the atrium

Temperature differences might occur in an atrium on for example a hot summer's day. This could lead to a stack effect whereas smoke is hindered from rising by a warmer layer of air in the atrium space. These conditions were simulated by prescribing an initial temperature that increased by height in the atrium, starting at 25° Celsius at the ground and ending with a temperature of 50° Celsius just below the ceiling of the atrium.

5.4 Simulation of heat loss to a cold glass ceiling

Wall and roof materials might have an effect on how the smoke is cooled. If the materials that are used have a high thermal conductivity they will lead heat away at a faster rate. Also the thickness of these materials are a factor that needs to be considered, a thin glass roof doesn't provide much insulation. As the purpose of most atria is to create a light atmosphere in the shopping malls most of them are fitted with glass ceilings. This could lead to problems with smoke management in atria, especially in cold climates.

To investigate this issue simulations were done where the ceiling of the atrium was prescribed a temperature of 0° Celsius. As a worst case these conditions were simulated in the 180-meter long atrium model and in the 5000 m^2 square model. Smoke ventilation was not provided in these cases due to problems with prescribing boundary conditions between inside and outside the atrium space in FDS. However this study gives us an indicative idea of smoke behaviour in the atrium space and whether or not the smoke has sufficient buoyancy to be ventilated through ceiling vents if such were present.

5.5 Simulations with increased cell resolution

One issue related to the model that was considered important to investigate was whether or not the cell resolution of the model might have an effect on the results and if the solution might be considered grid independent. Simulations were run with refined meshes, either by having a finer grid for the entire control volume or refining the grid for areas near the fire or otherwise of particular interest. Narrowing the grid for particular areas was made possible by utilizing the “multi-block” function included in FDS v.3.1. Multi-blocking allows the user to prescribe different cell resolutions to different meshes/blocks of the model as illustrated in figure 5.1.

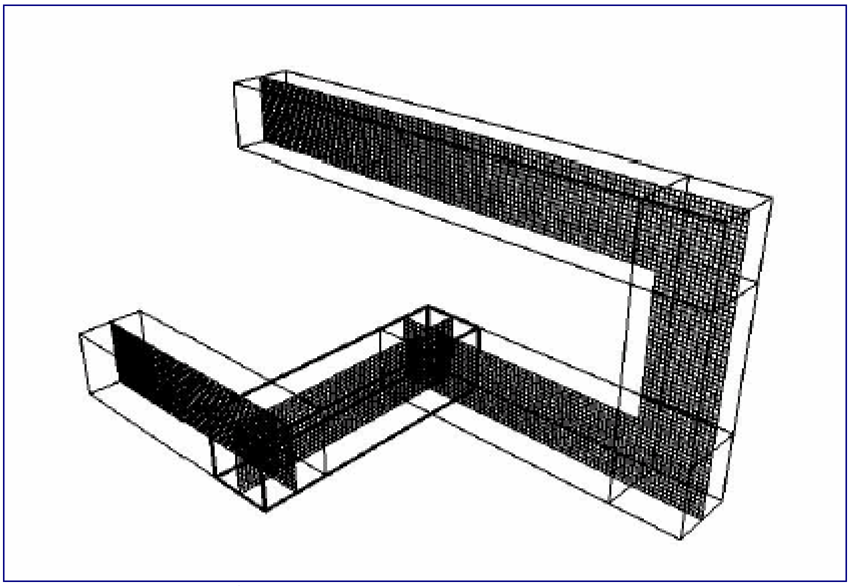


Figure 5.1, Example of multi-block geometry [4]

As FDS always uses the finest mesh configuration when multiple blocks interface the function could be used to redefine part of the existing atria model simply by prescribing a new block within the existing model. This was done in the 180-meter long atrium case where the mesh was refined close to the fire. The resolution was set at 0.25×0.25×0.25 meters for a block 10×5×4 meters around the fire and the rest of the model was kept at the original resolution.

6 Results of initial simulations

The results presented in this section are based on the assumtions and specifics given in section 4. The simulations are described in text and with the help of a few illustration. For a more detailed view of the results look to appendix C.

6.1 Long atrium

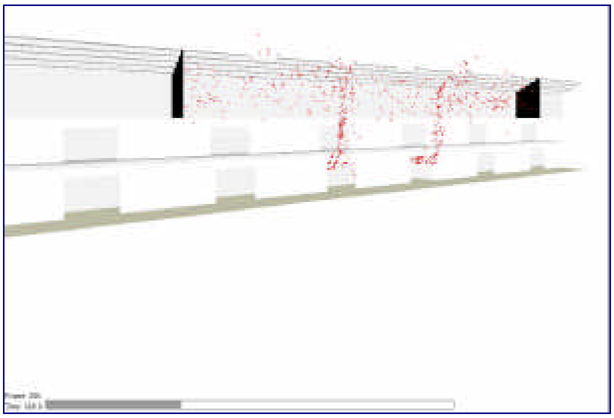


Figure 6.1, Particle tracings at t=300 seconds

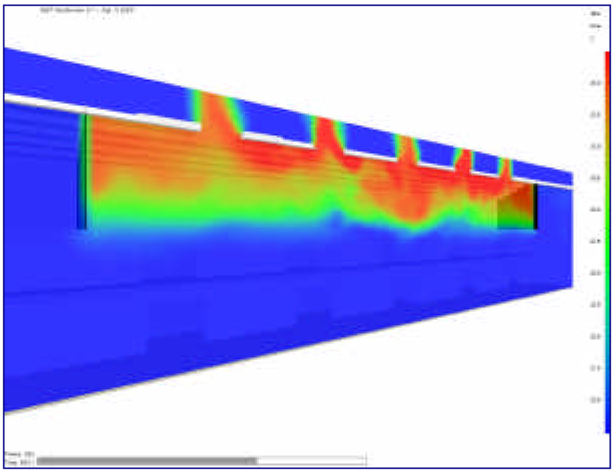


Figure 6.2, Temperature slice at t=600 seconds

In the initial simulation the smoke reservoir work as planned and the smoke is kept therein. A smoke layer with sufficient buoyancy forms and there is no problem ventilating the smoke through the smoke vents mounted in the ceiling. There is no spread of smoke, neither beyond the smoke barriers nor to the floors below the smoke reservoir, as illustrated in the figures presented to the left. Figure 6.1 shows the movement of particles from the enclosures illustrating shops and figure 6.2 shows the difference in the smoke layer temperature. Green represents a temperature of about 27 degrees Celsius and red represents temperatures in excess of 35 degrees Celsius.

The smoke layer stabilized within the barriers throughout the simulation.

No spilling of smoke was observed beyond the smoke barriers.

The average temperature of the smoke layer was in the order of 30 to 35 degrees Celsius. This temperature is considered sufficient to provide the smoke with enough thermal buoyancy to avoid problems with stratification.

The flow through the smoke vents had a maximum velocity of about 2.5 meter/second.

For further pictures illustrating the smoke movement in detail at different times in the simulation the reader is referred to appendix C.

6.2 Square atrium

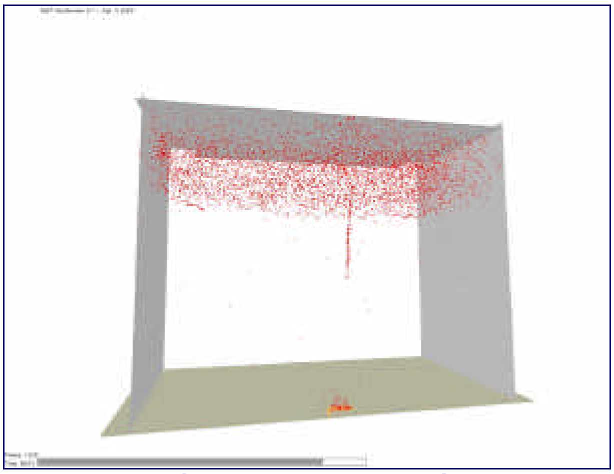


Figure 6.3, Particle tracings at t=600 seconds

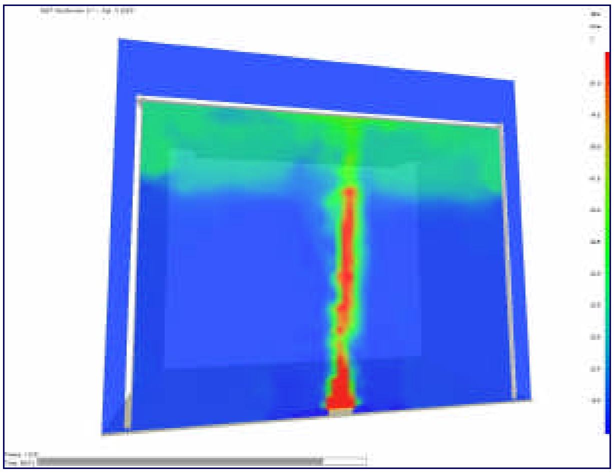


Figure 6.4, Temperature slice at t=600 seconds

In the initial simulation of the square shaped smoke reservoir the smoke management worked as planned. A smoke layer with sufficient buoyancy forms and there is no problem ventilating the smoke through the smoke vents mounted in the ceiling. There is no spread of smoke, to the floors below the smoke reservoir, as illustrated in the figures presented to the left.

The same situation occurs here as in the previous test. Smoke is kept at ceiling height and the smoke management systems are allowed to work with the help from the buoyancy of the smoke.

No loss of buoyancy was observed throughout the time of the simulation. The average temperature of the smoke layer was in the order of 50 degrees Celsius. This temperature is considered sufficient to provide the smoke with enough thermal buoyancy to avoid problems with stratification.

The flow through the smoke vents had a maximum velocity of about 2.5 meter/second.

For further pictures illustrating the smoke movement at different times in the simulation the reader is referred to Appendix C.

6.3 Summary of initial simulations

The initial simulations indicate that in normal conditions these arbitrary limits are set at a level where a smoke management system is not prevented from working as intended. As suspected the limits are conservative and just as these types of limits should be.

The results from these simulations shows that the limits mentioned in international reference literature are likely to be stretched rather than shrunk with the help of analytic fire safety design. However there are a number of tests that need to be added to the analysis before one can draw any certain conclusions. The further simulations in the following chapter shows the efforts to evaluate just which factors do influence the smoke movement in a negative manner.

7 Further simulations

After evaluating the initial simulations further simulations were conducted to both stretch the limmits and look at what parameters the model is most sensitive to. Further details are described with each run and in section 5. For details on the results the reader should look to appendix C.

7.1 Increased reservoir length

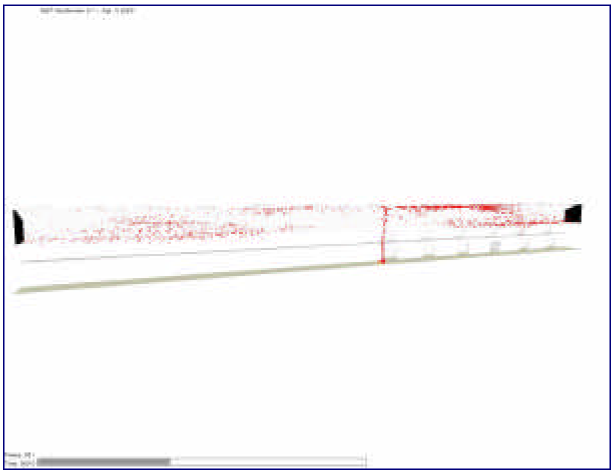


Figure 7.1, Particle tracings at t=300 seconds

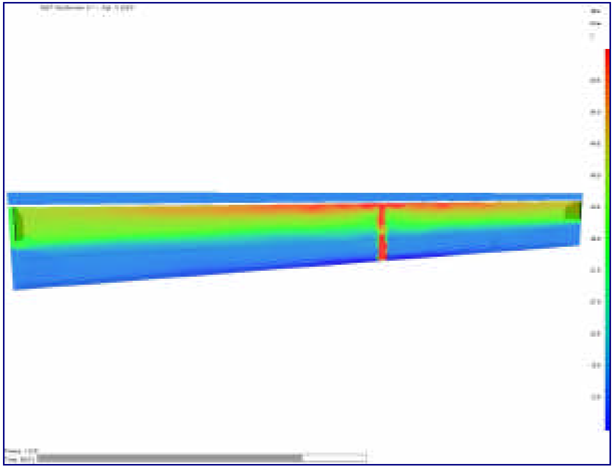


Figure 7.2, Temperature slice at t=600 seconds

In this run with a 90-meter long atrium as well as in the simulation with the 60-meter long atrium there was no problem containing the smoke at a high level, which is a necessity for the smoke management of such buildings.

The average temperature of the smoke layer was enough to create the buoyancy needed and the cooling from the ambient air was marginal. For further pictures illustrating the smoke movement at different times in the simulation the reader is referred to Appendix C.

Even when the long atrium is stretched to three times as long as the original one the smoke preserves its buoyancy and forms a fairly uniform layer at the ceiling.

This shows in figure 7.1 and 7.2 where the temperature and particles produced are illustrated.

7.2 Increased reservoir area

The result from the simulations with an increased reservoir area the same was found as for the lengthened atrium. With the increased dimensions of the atrium

7.3 Heat loss to a cold glass ceiling

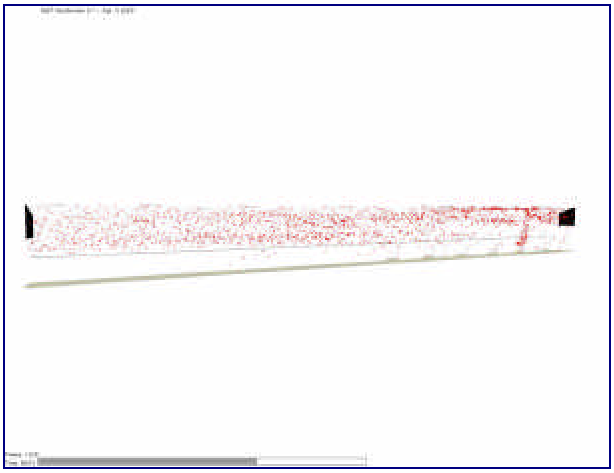


Figure 7.3, Particle file at t=600 seconds

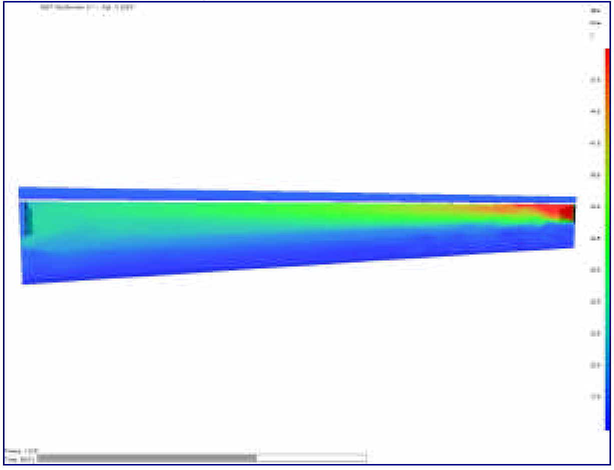


Figure 7.4, Temperature slice at t=600 seconds

The effect of heat loss to a cold surface has some effect to the stratification phenomena. One can tell that there is a finer mix between the gases produced by the fire and the ambient air. The smoke tends to lose both temperature and buoyancy the further away from the plume centreline it travels.

It is a fine difference between these runs and the once without the effect of the surface cooling. Though the fine difference it is still presence and can be seen in either or both of figures 7.3 and 7.4.

For more pictures from the run see appendix C.

7.4 Temperature differences within the atrium

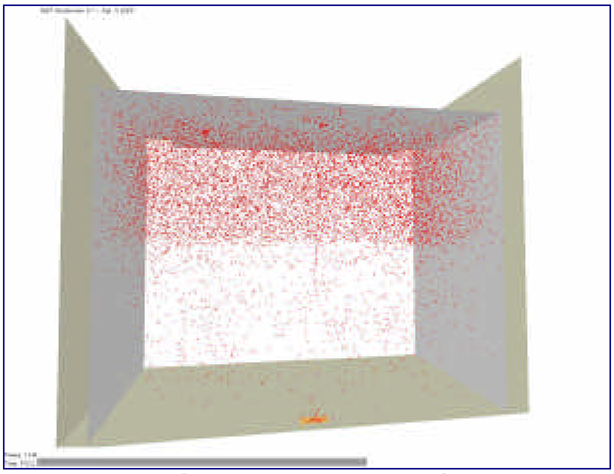


Figure 7.5, Particle tracings at t=600 seconds

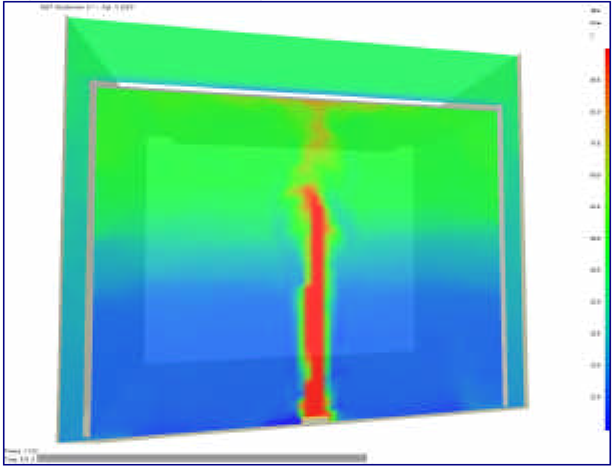


Figure 7.6, Temperature slice at t=600 seconds

This is the far most interesting run. After evaluating the effect of the atriums shape and size without any obvious signs of stratification this was the break through. With a temperature gradient throughout the whole height of the atrium the plume had great difficulties punching through the layers of hot ambient air at ceiling height.

The particle file in figure 7.5 shows the well-mixed case. There is still a large amount of smoke at the top of the atrium, but undoubtedly there is smoke at ground level as well.

Figure 7.6 does not give a clear picture of the smoke from the fire since it also includes the gradient prescribed at the start of the CFD run. The initial conditions can be seen on the side of the atrium which is considered as outside in these runs.

7.5 Increased cell resolution

As sensitivity analysis the resolution of the grid was altered to see how a finer grid than the one used would change the results.

The finer grid was prescribed throughout the volume of the atrium as described in 5.5

Comparing the results in this run with the ones from 7.1 shows no major changes in temperature, visibility or smoke movement with the diffrent resolutions. This indicates that the resolution used for the runs conducted in this project are well within the limits of what is reasonable.

The issue of grid idependence has not been sufficently investigated to claim that the results will be totally identical no matter the resolution of the grid. However the comparisons performed suggests that there is no major change in the results due to a finer grid. The difference are deifinitly not of the magnitude to cause any suspicion that phenomena might appear that are different from the ones observed in the run with the coarser grid resolution.

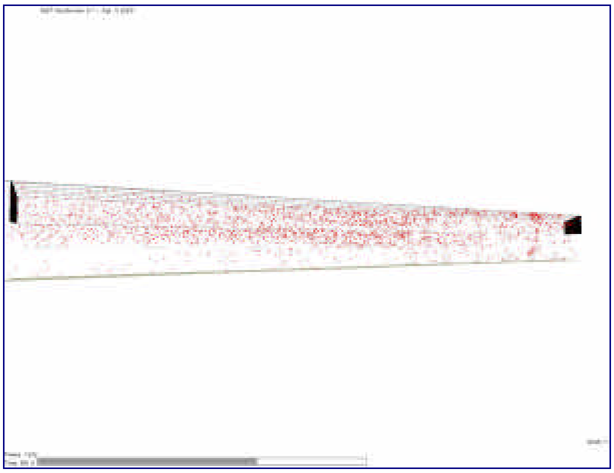


Figure 7.7, Particle file at t=600 seconds

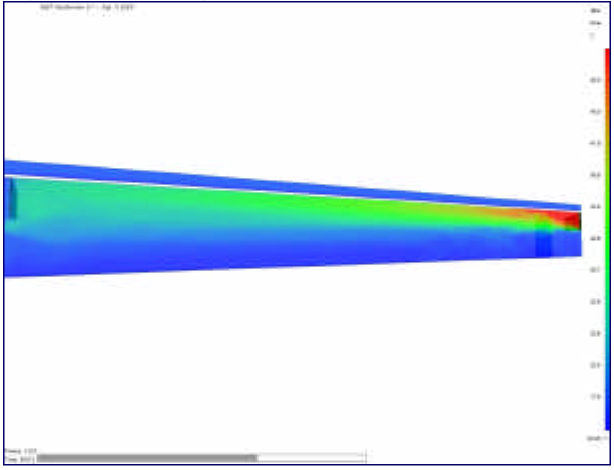


Figure 7.8, Temperature slice at t=600 seconds

7.6 Summary of further simulations

The further simulations done on the two different geometries shows that there are a number of factors one should take into account when designing smoke reservoirs. From the tests conducted it shows that the area and length of the reservoir is of less importance than e.g. the temperature gradient.

The factor that has the biggest impact for the stratification phenomena is first and foremost the temperature gradient in the atrium. This becomes most obvious in the high atriums where temperatures at ceiling height can reach fairly high temperatures compared to the ones at ground level.

Cooling from surrounding surfaces can also lessen the buoyancy of the smoke produced by the fire. This effect is secondary to the temperature gradient.

Though it is important to realize that just because the smoke reservoirs can be stretched beyond the limits mentioned in for instance BRE it doesn’t mean you can delete fire compartmentation out of the design. This is one factor in fire safety design and there are a number of others that should be taken into account.

A discussion on how to interpret and use the results from the report is held in the following chapter. Recommendations are given on how to use the CFD tools to get the most accurate and reliable results.

8 Conclusions and recommendations

There is a lot to say about the use and the design of smoke reservoirs. The aim in this project was to create a sort of tool for fire engineers, which could be used to limit the workload in future projects. Looking back one could say that the aim was set too high in relation to the resources and time which was available to the writers. Creating a model or a method of that kind would at least triple the amount of time and effort in this project. Unfortunately the time necessary has not been at hand. However this could be considered a case of ‘aiming for the stars and reaching the skies’.

A lot of experience has been gained by the authors through this project both with handling FDS as well as knowledge about smoke reservoirs. Hopefully some of the conclusions drawn from the work performed (especially the problems and difficulties) in this project can be useful for other users of FDS, both in design of smoke reservoirs and using of FDS as a design tool in general.

There are a lot of issues concerning CFD-modelling and time is always an issue when it’s used as a design tool. Even though this project maybe has not examined all issues to full depth or in some cases only scratched the surface the evaluations has gone above and beyond the usual approach in the consulting business where the use of CFD-modelling is widespread.

As for the result of this project the outcome did not become what initially was intended, but what research can be defined before the results are at hand? As the work progressed it became clear that a method for using CFD as a design tool for smoke management in atria was a more realistic goal than using CFD to decide on a general model for the size of smoke reservoirs as local conditions and specific building geometries can vary to an infinity.

The recommendations used today are inflexible and put restrictions on how atria can be designed. A method for using CFD in the design of smoke reservoirs can be very useful in creating cost-effective and architectonically attractive design of malls and atria without an excess of fire safety measurements. If an alternative design of the smoke reservoirs within an atrium is suggested this should always be verified with a CFD-simulation of a fire scenario for the specific case. To simplify this process this project has resulted in a number of points to consider when using CFD as a design tool, these are:

- Always include the local meteorological conditions at the location in your model
- Define a fine grid for areas close to the fire and ensure that the solution is grid independent
- If possible use ventilation consultants data on the internal climate since initial temperature differences can have a more extensive effect on the behaviour of smoke than the physical size of the reservoir.
- Increase the size with caution since the effectiveness of smoke vents decrease as the smoke is cooled.
- Since there are always a certain amount of uncertainties involved in fire modelling a margin of error must always be included in the calculations.

Computational Fluid Dynamics can be a useful tool to verify alternative solutions where the sizes of the reservoirs are larger than the recommended. However, CFD is not useful as an iterative tool for smoke management solutions because of the long simulation times. One should use simpler calculation methods to decide a preliminary design of the smoke management in the atria and then verify this solution with the help of CFD-modelling.

It is important to remember that these recomendations are the results of the modelling done within this project and under the given circumstances therein. It has not been possible to analyze the different parameters to the extant necessary to completely overlook them in the design of these smokereservoirs.

9 References

[1] Morgan H.P, Gardner J.P, *Design principles for smoke ventilation in enclosed shopping centres*, BRE Report BR 186

[2] *Boverkets Byggregler, BBR*, BFS 1993:57 med ändringar till och med 2002:19, Boverket 2002.

[3] Kevin B. McGrattan et.al., Fire Dynamics Simulator (Version 3) – Technical Reference Guide NISTIR 6783, 2002 Edition

[4] Kevin B. McGrattan et.al., *Fire Dynamics Simulator (Version 3) – User's Guide*, NISTIR 6784, 2002 Edition

[5] Davis W.D, Notarianni K.A and McGrattan K.B, *Comparison of fire model predictions with experiments conducted in a hangar with a 15 meter ceiling*, Technical report NIST internal report (NISTIR 5927), National Institute of Standards and Technology, Gaithersburg, Maryland 20899, 1996.

[6] J.E. Floyd. “Comparison of CFAST and FDS for fire simulation with the HDR T51 and T52 tests”. Technical report NIST internal report (NISTIR 6866), National Institute of Standards and Technology, Gaithersburg, Maryland 20899, 2002.

[7] Brenig, H.W., et al “First evaluation of the Fire Computer Codes Applied to the Analysis of the HDR Fire Experiments (in german)” PDHR Working Report 5.093/86. Karlsruhe, Germany: Project HDR, Nuclear Center Karlsruhe, 1989.

[8] Discussions with Bo Svensson, ScheiwillerSvensson Architechts, Stockholm April 2003

[9] *Ansys homepage on CFX*, http://www-waterloo.ansys.com/cfx/, 2004.

[10] *Cranfield University homepage on Sofie*, http://www.cranfield.ac.uk/sme/sofie/, 2004.

[11] NIST homepage on FDS and Smokeview, http://fire.nist.gov/fds/, 2004

[12] Karlsson B, Quintere JG., *Enclosure Fire Dynamics*, CRC Press 1999

[13] Bengt Hägglund et.al., *Effects on inlets on natural fire vents – an experimental study* , FOA-R—96-0024-2.4—SE, National Defense Research Establishment – Department of Weapons and Protection, Stockholm, Sweden, May 1996

[14] Morgan H.P, Marshall N.R, *Smoke hazards in covered multi-level shopping malls: An experimentally based theory for smoke production*, BRE current paper CP 48/75

Appendix A – Indata file

The input data is where the geometry of the enclosure, the fire and the ambient conditions is set. Below is an example of a file used in this project. Of course there are a lot more commands than the ones presented here. However this example gives an idea on what a basic indata file would look like.

```
& HEAD CHID= 'Atrium3',
  TITLE= 'Atrium3'/
```

- This is where the title of the run is described.

```
& TIME TWFIN= 900/
```

- This run will continue for 15 minutes.

```
& MISC NFRAMES= 900,
  SURF_DEFAULT= 'GYPSUM BOARD',
  DATABASE= 'c:\nist\fds\database3\database3.data',
  TMPA= 20,
  TMPO= 20,
```

- This indicates the number of frames.
- Here gypsum is the default surface material.
- The database file used.
- Ambient temperature
- Outside temperature

```
& PDIM
  XBAR= 90,
  YBAR= 20,
  ZBAR= 20/
```

- The grid length in the x-direction.
- The grid length in the y-direction.
- The grid length in the z-direction.

```
& GRID
  IBAR= 180,
  JBAR= 40,
  KBAR= 40,/
```

- The number of cells in the x-direction.
- The number of cells in the y-direction.
- The number of cells in the z-direction.

```
& SURF ID= 'Burner',
  HRRPUA= 625,
  TAU_Q= -326,
  PARTICLES=.TRUE.,
  PARTICLE_COLOR='RED'  /Fast 2MW fire
```

- Defining the surface of the fire.
- Maximum heat release rate per area unit.
- Defining the Fast at^2 fire.
- Telling FDS to trace particles from the fire

```
& OBST XB = 44, 46, 1, 3, 0.0, 0.5,
  SURF_IDS = 'Burner', 'Burner', 'INERT'/
```

- Describing the fire block (x, x_1, y, y_1, z, z_1)
- Defining the sides of the ‘fire block’.

```
& VENT CB= 'XBAR', SURF_ID='OPEN' /
& VENT CB= 'XBAR0', SURF_ID='OPEN' /
& VENT CB= 'YBAR', SURF_ID='OPEN' /
& VENT CB= 'YBAR0', SURF_ID='OPEN' /
& VENT CB= 'ZBAR', SURF_ID='OPEN' /
```

- Defines if the sides of the volume are open to the outside or not.

```
& OBST XB= 0, 90 , 0 , 0.5 , 0 , 10 /
& OBST XB= 0 , 90 , 19.5 , 20 , 0 , 10 /
& OBST XB= 0 , 90 , 0 , 5 , 5 , 5.5 /
& OBST XB= 0 , 90 , 15 , 20 , 5 , 5.5 /
& OBST XB= 0 , 90 , 4.5 , 5 , 10 , 15 /
& OBST XB= 0 , 90 , 15 , 15.5 , 10 , 15 /
& OBST XB= 0 , 90 , 5 , 6 , 15 , 15.5 /
& OBST XB= 0 , 90 , 6 , 7 , 15.5 , 16 /
& OBST XB= 0 , 90 , 7 , 8 , 16 , 16.5 /
& OBST XB= 0 , 90 , 8 , 9 , 16.5 , 17 /
& OBST XB= 0 , 90 , 11 , 12 , 17 , 16.5 /
& OBST XB= 0 , 90 , 12 , 13 , 16.5 , 16 /
& OBST XB=……etc.
```

- Defines the blocks, which together forms the enclosure. In a complex geometry there will be thousands of blocks like these.

```
&  SLCF PBX= 45 ,QUANTITY=  'VELOCITY'  /
&  SLCF PBY= 10 ,QUANTITY=  'VELOCITY'  /
```

- Slice files showing absolute velocity in the x- and y-plane.

```
&  SLCF PBX= 16 ,QUANTITY=  'visibility'  /
&  SLCF PBX= 45 ,QUANTITY=  'visibility'  /
&  SLCF PBX= 74 ,QUANTITY=  'visibility'  /
```

- Slice files showing visibility in the x-plane.

```
&  SLCF PBZ= 1  ,QUANTITY=  'visibility'  /
&  SLCF PBZ= 3  ,QUANTITY=  'visibility'  /
&  SLCF PBZ= 5 ,QUANTITY=  'visibility'  /
&  SLCF PBZ= 7 ,QUANTITY=  'visibility'  /
&  SLCF PBZ= 9 ,QUANTITY=  'visibility'  /
&  SLCF PBZ= 10 ,QUANTITY=  'visibility'  /
&  SLCF PBZ= 11 ,QUANTITY=  'visibility'  /
&  SLCF PBZ= 12 ,QUANTITY=  'visibility'  /
&  SLCF PBZ= 14 ,QUANTITY=  'visibility'  /
```

- Slice files showing visibility in the z-plane.

```
&  SLCF PBY= 5.5 ,QUANTITY=  'visibility'  /
&  SLCF PBY= 10.5 ,QUANTITY=  'visibility'  /
&  SLCF PBY= 14.5 ,QUANTITY=  'visibility'  /
```

- Slice files showing visibility in the y-plane.

```
&  SLCF PBX= 16 ,QUANTITY=  'TEMPERATURE'  /
&  SLCF PBX= 45 ,QUANTITY=  'TEMPERATURE'  /
&  SLCF PBX= 74 ,QUANTITY=  'TEMPERATURE'  /
```

- Slice files showing temperature in the x-plane.

```
&  SLCF PBZ= 1  ,QUANTITY=  'TEMPERATURE'  /
&  SLCF PBZ= 3  ,QUANTITY=  'TEMPERATURE'  /
&  SLCF PBZ= 5 ,QUANTITY=  'TEMPERATURE'  /
&  SLCF PBZ= 7 ,QUANTITY=  'TEMPERATURE'  /
&  SLCF PBZ= 9 ,QUANTITY=  'TEMPERATURE'  /
&  SLCF PBZ= 10 ,QUANTITY=  'TEMPERATURE'  /
&  SLCF PBZ= 11 ,QUANTITY=  'TEMPERATURE'  /
&  SLCF PBZ= 12 ,QUANTITY=  'TEMPERATURE'  /
&  SLCF PBZ= 14 ,QUANTITY=  'TEMPERATURE'  /
```

- Slice files showing temperature in the z-plane.

```
&  SLCF PBY= 5.5 ,QUANTITY=  'TEMPERATURE'  /
&  SLCF PBY= 10.5 ,QUANTITY=  'TEMPERATURE'  /
&  SLCF PBY= 14.5 ,QUANTITY=  'TEMPERATURE'  /
```

- Slice files showing temperature in the z-plane.

```
&  ISOF QUANTITY='TEMPERATURE' ,
   VALUE(1)=30, VALUE(2)=50  ,VALUE(3)=150  /
```

- Isosurface showing temperature (30, 50 and 150°C)

Appendix B, Smoke movement calculations

This section describes the equations used to decide the area of smoke vents. These formulae are included in a spreadsheet model where the required ventilation in each case can be calculated in an iterative fashion.

The fire is assumed to form an axisymmetric plume. Accordingly, the mass entrainment, M, into the plume can be described by the following equation (equation 14 in NFPA 92B):

$$M = 0.071 {Q_p}^{1/3} z^{5/3} + 0.0018 Q_p$$

Where: M = mass entrainment into the plume (kg/s)
Q_p = convective portion of heat release rate, 70% of actual heat release rate (kW)
Z = smoke layer height above base of fire/floor (m)

When the limiting flame height (z_1) is greater than the height to the smoke layer, i.e. $z_1 \geq z$, the entrainment must be calculated with the following equation (equation 15 in NFPA 92B):

$$M = 0.032 {Q_p}^{3/5} z [1.1]$$

Where: M = mass entrainment into the plume (kg/s)
Q_p = convective portion of heat release rate, 70% of actual heat release rate (kW)
z = smoke layer height above base of fire (m)

The limiting flame height can be calculated by using the formula below (equation 13 in NFPA 92B).

$$z_1 = 0.166 {Q_p}^{2/5} [1.2]$$

The elapsed time at which the smoke free layer is at a height z is obtained by solving the following differential equation (equation 6.1 in CIBSE TM 19:1995):

$$r_o A_f \frac{dz}{dt} + M + \frac{Q_p}{T_0 C_p} = 0 [2]$$

Where: ρ_0 = density of ambient air (kg/m^3)
A_f = floor area of room (m^2)
z = height above base of fire (m)
M = mass flow of entrained air (kg/s)
Q_p = convective portion of heat release rate (kW)
T_0 = ambient air temperature (K)
C_p = specific heat capacity of air (kJ/kgK)

The use of a finite difference method, which is explained below, can solve the above differential equation.

Mass entrainment rate is calculated with a time step of ten seconds. It is assumed that all the mass and energy from the fire plume enters the smoke layer. The mass of the smoke layer, M_{st}, is therefore equal;

$$M_{st} = M_{st-1} + M \Delta t [3]$$

Where: M_{st-1} = mass of the smoke layer from the previous time step (kg)
M = mass entrainment rate from equation [1] (kg/s)
Δt = time step used in the spreadsheet analysis = 10 s
M_{st} = mass of the smoke layer at the end of the time step (kg)

When natural smoke ventilation, i.e. ventilation due to differences in temperature, is installed in the building, equation [3] must be written as follows:

$$M_{st} = M_{st-1} + M\Delta t - M_{nat}\Delta t [3.1]$$

$$M_{nat} = \frac{C_d A_{vo} r_0 \left[2g(h - z_t)(T_{st} - T_0)T_0\right]^{1/2}}{T_{st}^{1/2}\left[T_{st} + (A_{vo}/A_{vi})^2 T_0\right]^{1/2}} [3.2]$$

Where: M_{nat} = mass flow of the vented smoke (kg/s), se equation [3.2].

The mass flow of the vented smoke can be calculated with equation 6.14 in CIBSE TM 19:1995.

Where: C_d = discharge coefficient = 0.7
A_{vo} = outlet ventilation area (m^2)
A_{vi} = inlet ventilation area (m^2)
ρ_0 = density of ambient air (kg/m^3)
g = acceleration due to gravity (m/s^2)
h = floor-to-ceiling height of room (m)
z_t = height from floor to smoke layer in the beginning of the time step (m), from equation [9]

T_{st} = average temperature of the smoke layer in the beginning of the time step (K), from equation [5]
T_0 = ambient temperature (K)

The time from when the fire starts until the natural ventilation are fully operating can be determined by comparing the calculated smoke obscurity in the spreadsheet with the sensitivity of the smoke detection systems provided by manufacturers.

The heat content of the smoke layer, H_t, at each time step is equal to;

$$H_t = H_{t-1} + Q_p\Delta t - M_{nat}C_p\Delta T_{st-0}\Delta t [4]$$

Where: H_{t-1} = heat content of the smoke layer in the previous time step (kJ)
Q_p = convective portion of heat release rate (kW)
Δt = time step used in the spreadsheet analysis = 10s
H_t = heat content of the smoke layer at the end of the time step (kJ)
M_{nat} = mass flow of the vented smoke (kg/s)
C_p = specific heat capacity of air (kJ/kgK)
ΔT_{st-0} = temperature difference between smoke layer and ambient air (K), from equation [5]

The average temperature of the layer can be determined by using equation [5]:

$$T_{st} = T_0 + \frac{H_t}{M_{st}C_p} [5]$$

Where: T_{st} = temperature of smoke layer at the end of the time step (K)

T_0 = ambient temperature of the smoke layer (K)
H_t = heat content of the layer (kJ)
M_{st} = total mass of the smoke layer at the relevant time step (kg)
C_p = specific heat capacity of air = 1 kJ/kgK

The average plume temperature can be determined by using equation 5.17 in CIBSE TM 19:1995:

$$T_{mt} = T_0 + \frac{Q_p}{MC_p} [6]$$

Where:
T_{mt} = average plume temperature at the relevant time step (K)
T_0 = ambient temperature of the smoke layer (K)
Q_p = convective portion of heat release rate (kW)
M = mass entrainment rate from equation [1] (kg/s) at the relevant time step
C_p = specific heat capacity of air = 1 kJ/kgK

The volume of the smoke layer can be calculated by using equation [7] where the volume flow is given by equation 5.19 in CIBSE TM 19:1995:

$$V_t = V_{t-1} + \left(\frac{M_{st}}{\rho_0} + \frac{Q_p}{\rho_0 T_0 C_p}\right)\Delta t - \frac{M_{nat}}{\rho_{layer}} \Delta t [7]$$

Where:
V_t = volume of the smoke layer at the time step (m^3)
V_{t-1} = volume of the smoke layer at the previous time step (m^3)
M_{st} = mass flow of entrained air (kg)
Q_p = convective portion of heat release rate (kW)
ρ_0 = density of ambient air (kg/m^3)
T_0 = ambient temperature (K)
C_p = specific heat capacity of air = 1 kJ/kgK
Δt = time step used in the spreadsheet analysis = 10 s
M_{nat} = mass flow of the vented smoke (kg/s)
ρ_{layer} = average density of smoke layer (kg/m^3)

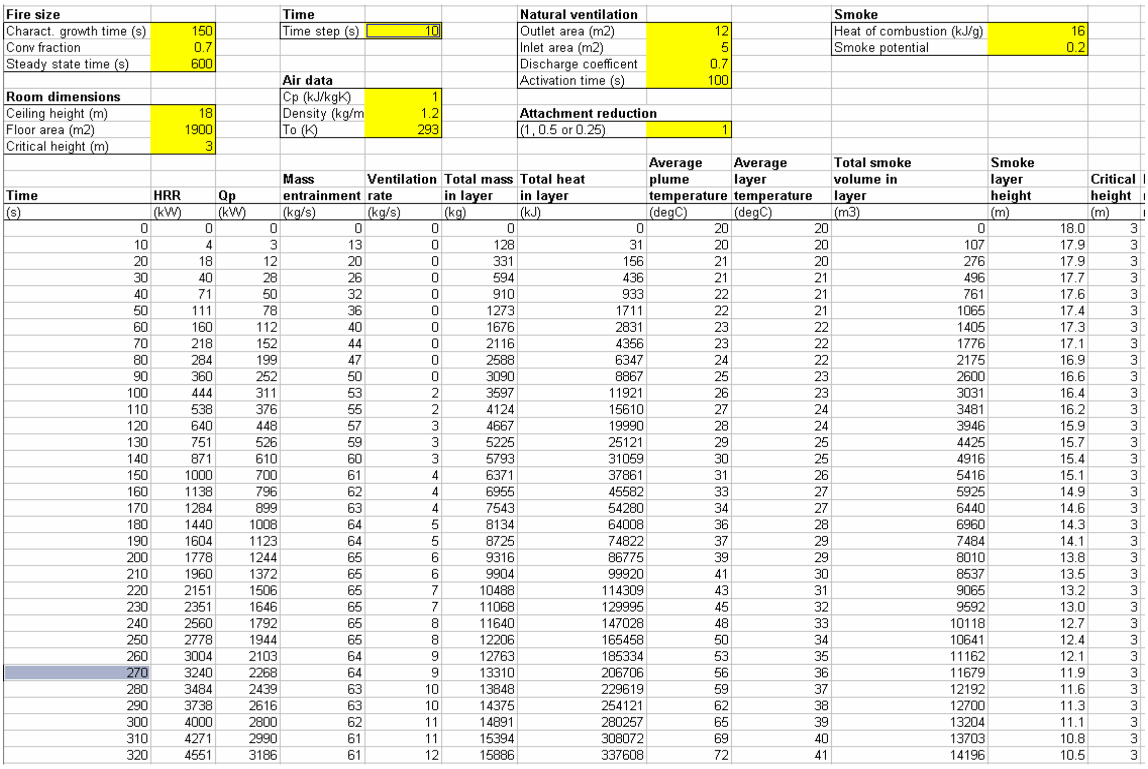
The depth of the smoke layer at the relevant time step (D_t) is equal to the volume (V_t) at that time step divided by the floor area (A_f) of the room of concern, i.e.

$$D_t = \frac{V_t}{A_f} [8]$$

The height to the smoke layer (z_t) is equal to the clear height (h) of the room minus the smoke layer depth at that particular time step, i.e.

$$z_t = h - D_t [9]$$

The smoke layer height at the end of each time step is used as the input smoke layer height for the next time step.



Fire size

Charact. growth time (s)	150
Conv fraction	0.7
Steady state time (s)	600

Room dimensions

Ceiling height (m)	18
Floor area (m2)	1900
Critical height (m)	3

Time

Time step (s)	10

Air data

Cp (kJ/kgK)	1
Density (kg/m	1.2
To (K)	293

Natural ventilation

Outlet area (m2)	12
Inlet area (m2)	5
Discharge coefficent	0.7
Activation time (s)	100

Attachment reduction

(1, 0.5 or 0.25)	1

Smoke

Heat of combustion (kJ/g)	16
Smoke potential	0.2

Time	HRR	Qp	Mass entrainment	Ventilation rate	Total mass in layer	Total heat in layer	Average plume temperature	Average layer temperature	Total smoke volume in layer	Smoke layer height	Critical height
(s)	(kW)	(kW)	(kg/s)	(kg/s)	(kg)	(kJ)	(degC)	(degC)	(m3)	(m)	(m)
0	0	0	0	0	0	0	20	20	0	18.0	3
10	4	3	13	0	128	31	20	20	107	17.9	3
20	18	12	20	0	331	156	21	20	276	17.9	3
30	40	28	26	0	594	436	21	21	496	17.7	3
40	71	50	32	0	910	933	22	21	761	17.6	3
50	111	78	36	0	1273	1711	22	21	1065	17.4	3
60	160	112	40	0	1676	2831	23	22	1405	17.3	3
70	218	152	44	0	2116	4356	23	22	1776	17.1	3
80	284	199	47	0	2588	6347	24	22	2175	16.9	3
90	360	252	50	0	3090	8867	25	23	2600	16.6	3
100	444	311	53	2	3597	11921	26	23	3031	16.4	3
110	538	376	55	2	4124	15610	27	24	3481	16.2	3
120	640	448	57	3	4667	19990	28	24	3946	15.9	3
130	751	526	59	3	5225	25121	29	25	4425	15.7	3
140	871	610	60	3	5793	31059	30	25	4916	15.4	3
150	1000	700	61	4	6371	37861	31	26	5416	15.1	3
160	1138	796	62	4	6955	45582	33	27	5925	14.9	3
170	1284	899	63	4	7543	54280	34	27	6440	14.6	3
180	1440	1008	64	5	8134	64008	36	28	6960	14.3	3
190	1604	1123	64	5	8725	74822	37	29	7484	14.1	3
200	1778	1244	65	6	9316	86775	39	29	8010	13.8	3
210	1960	1372	65	6	9904	99920	41	30	8537	13.5	3
220	2151	1506	65	7	10488	114309	43	31	9065	13.2	3
230	2351	1646	65	7	11068	129995	45	32	9592	13.0	3
240	2560	1792	65	8	11640	147028	48	33	10118	12.7	3
250	2778	1944	65	8	12206	165458	50	34	10641	12.4	3
260	3004	2103	64	9	12763	185334	53	35	11162	12.1	3
270	3240	2268	64	9	13310	206706	56	36	11679	11.9	3
280	3484	2439	63	10	13848	229619	59	37	12192	11.6	3
290	3738	2616	63	10	14375	254121	62	38	12700	11.3	3
300	4000	2800	62	11	14891	280257	65	39	13204	11.1	3
310	4271	2990	61	11	15394	308072	69	40	13703	10.8	3
320	4551	3186	61	12	15886	337608	72	41	14196	10.5	3

Figure B1.1 Example of spreadsheet calculation

Appendix C - Results

C.1 Initial simulation -The long atrium

Particles

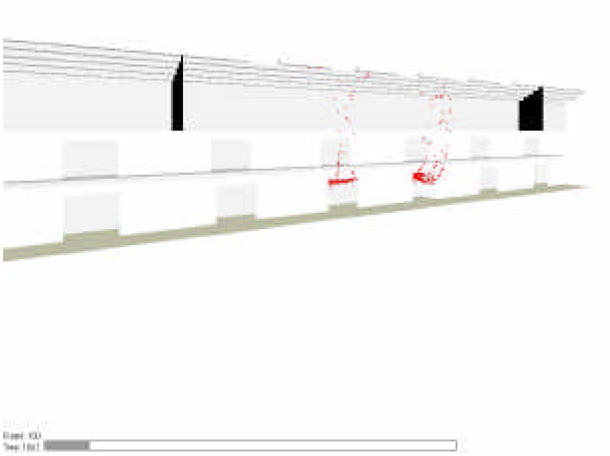


Figure C1.1, Particulate tracings at t=100 seconds

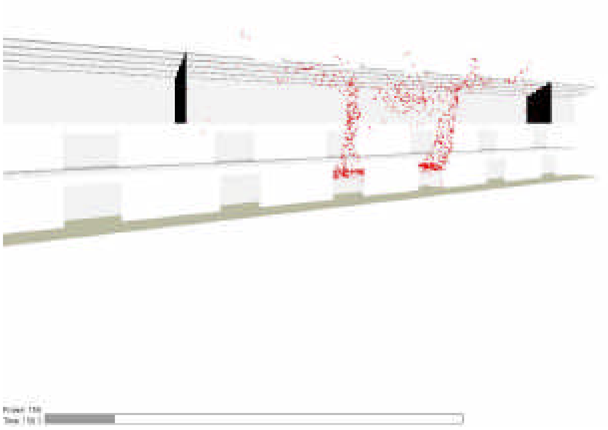


Figure C1.2, Particulate tracings at t=150 seconds

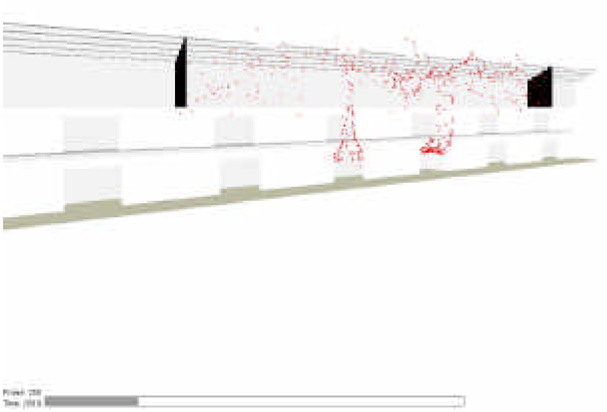


Figure C1.3, Particulate tracings at t=200 seconds

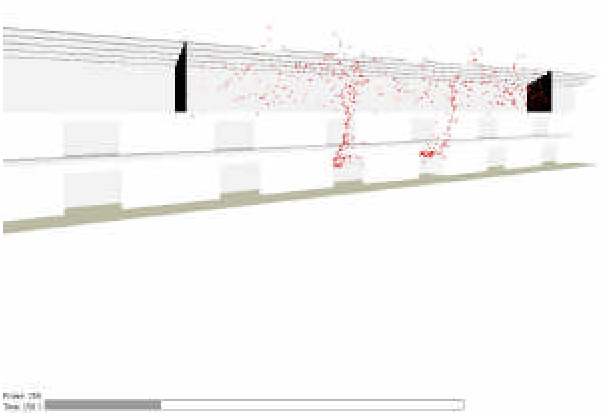


Figure C1.4, Particulate tracings at t=250 seconds

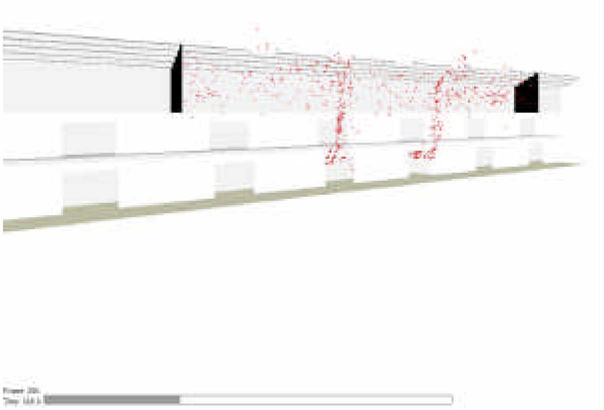


Figure C1.5, Particulate tracings at t=300 seconds

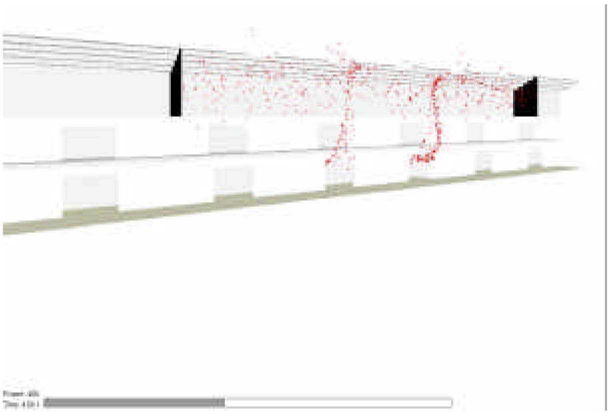


Figure C1.6, Particulate tracings at t=400 seconds

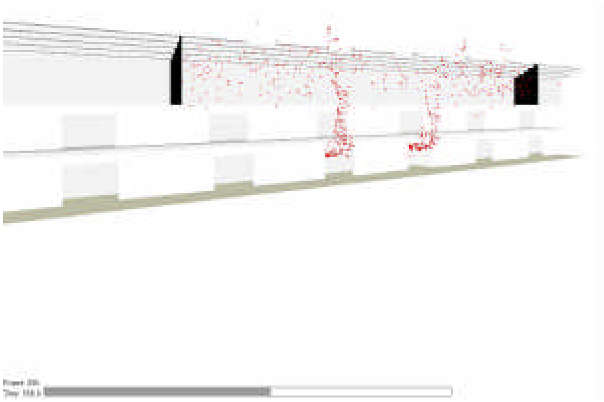


Figure C1.7, Particulate tracings at t=500 seconds

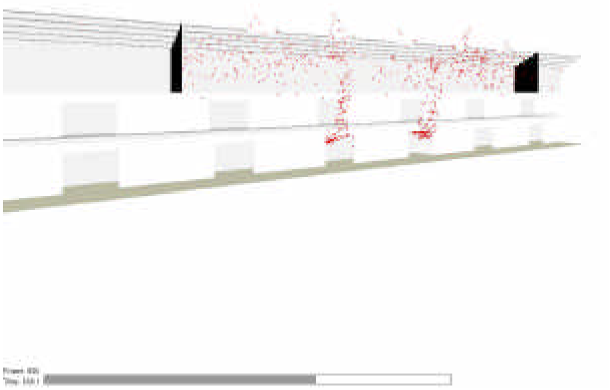


Figure C1.8, Particulate tracings at t=600 seconds

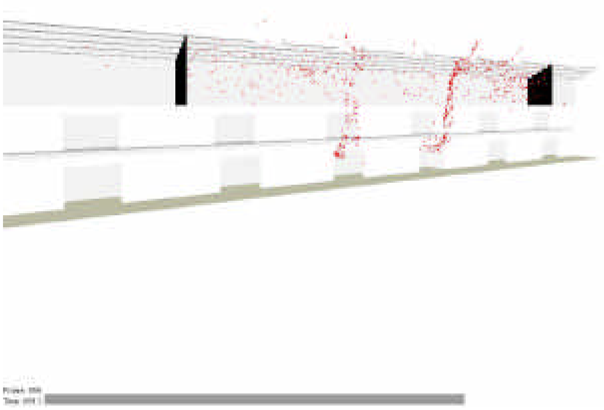


Figure C1.9, Particulate tracings at t=900 seconds

Temperature

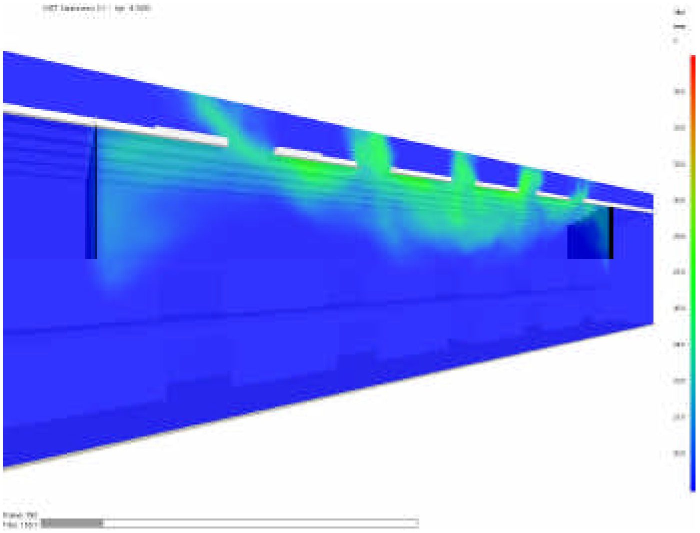


Figure C1.10, Temperature slice at t=150 seconds

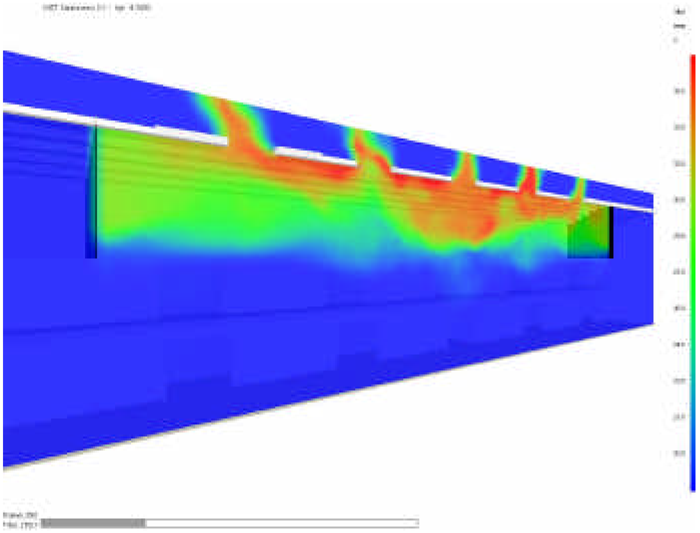


Figure C1.11, Temperature slice at t=250 seconds

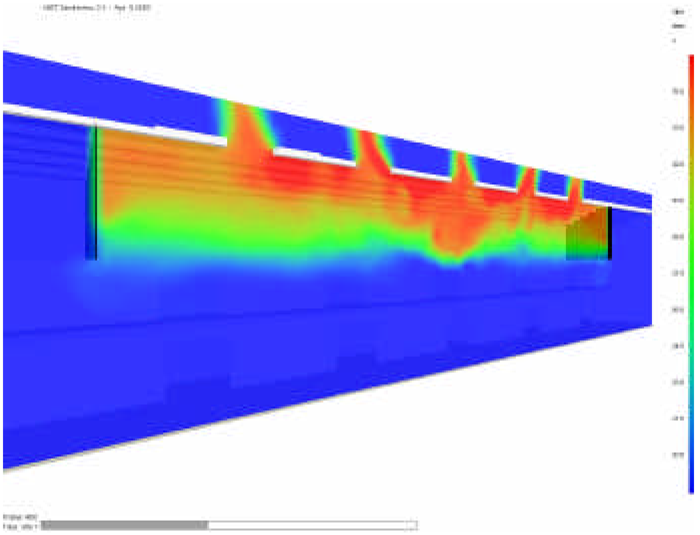


Figure C1.12, Temperature slice at t=400 seconds

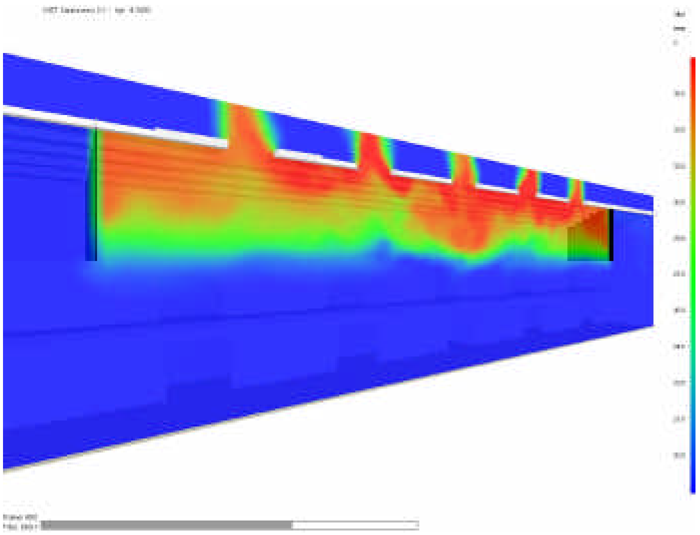


Figure C1.13, Temperature slice at t=600 seconds

C2. Initial simulation - The square atrium

Particles

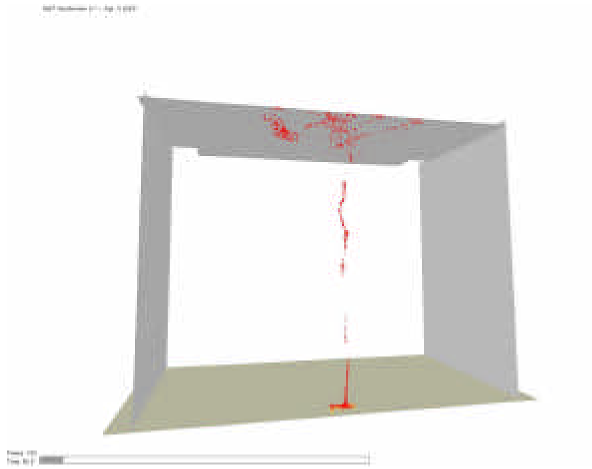


Figure C2.1, Particulate tracings at t=50 seconds

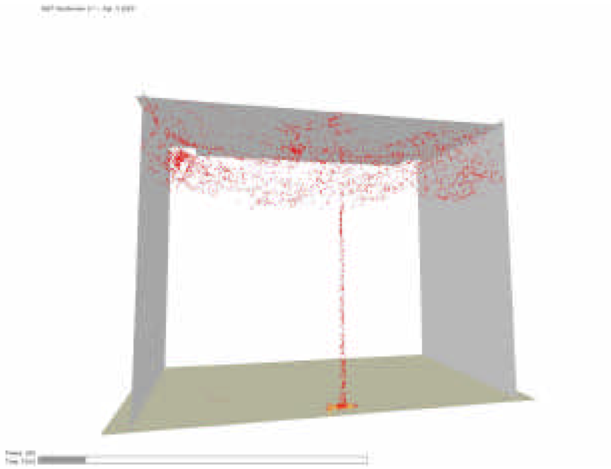


Figure C2.2, Particulate tracings at t=100 seconds

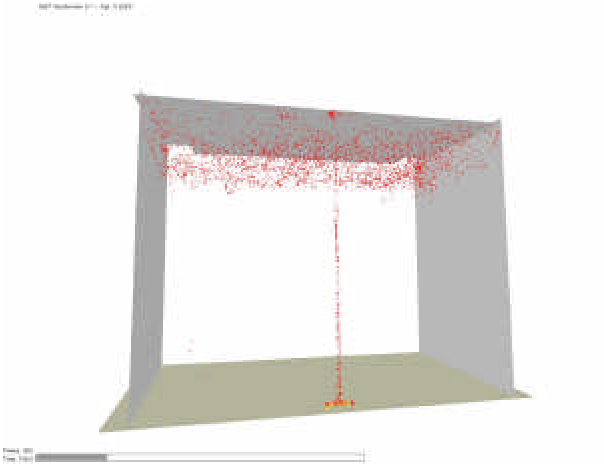


Figure C2.3, Particulate tracings at t=150 seconds

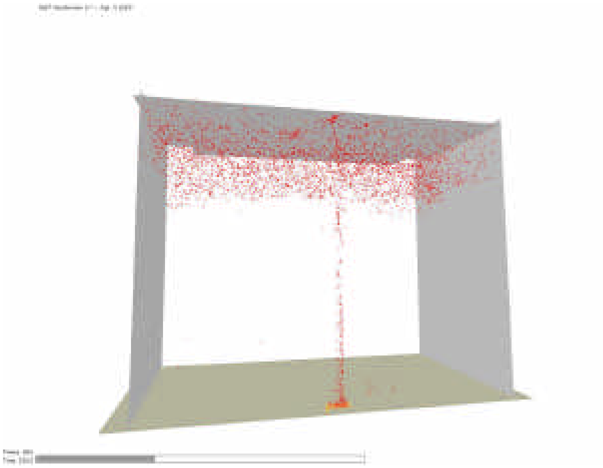


Figure C2.4, Particulate tracings at t=250 seconds

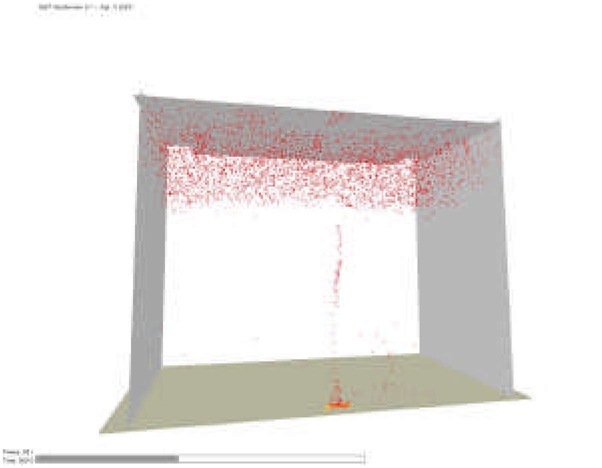


Figure C2.5, Particulate tracings at t=300 seconds

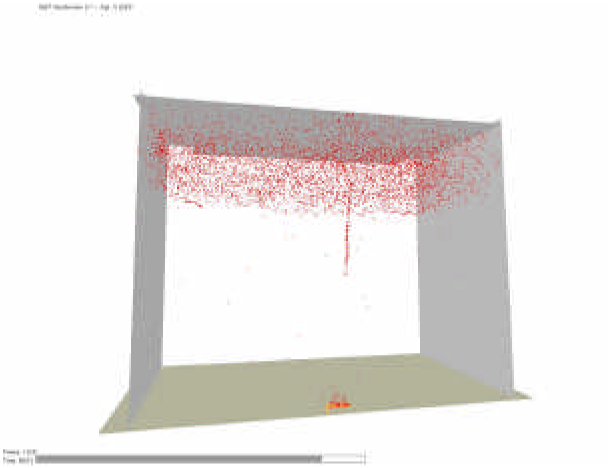


Figure C2.6, Particulate tracings at t=600 seconds

Temperature

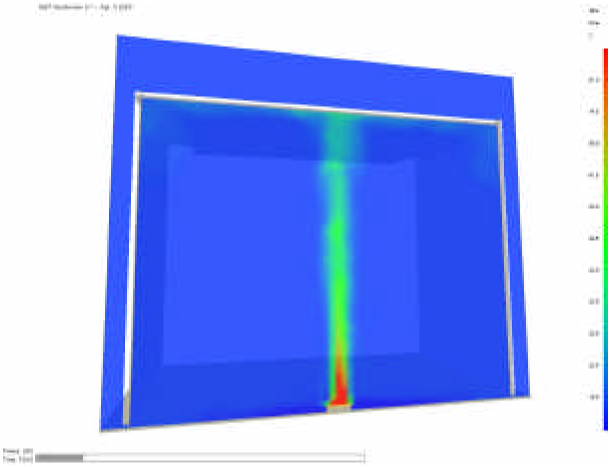


Figure C2.7, Temperature slice at t=100 seconds

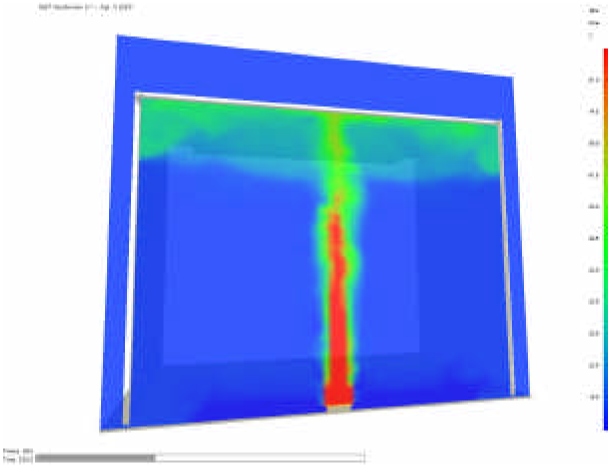


Figure C2.8, Temperature slice at t=250 seconds

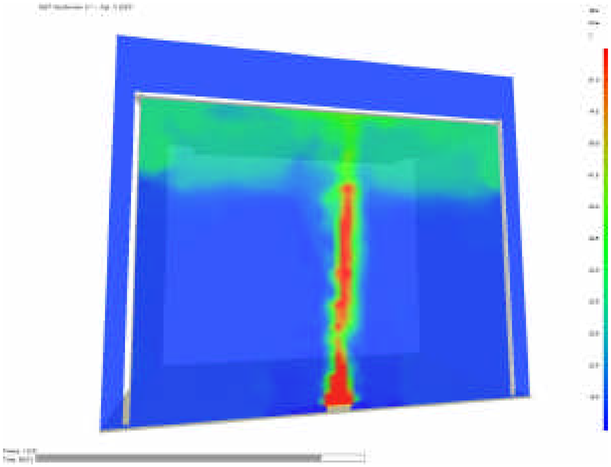
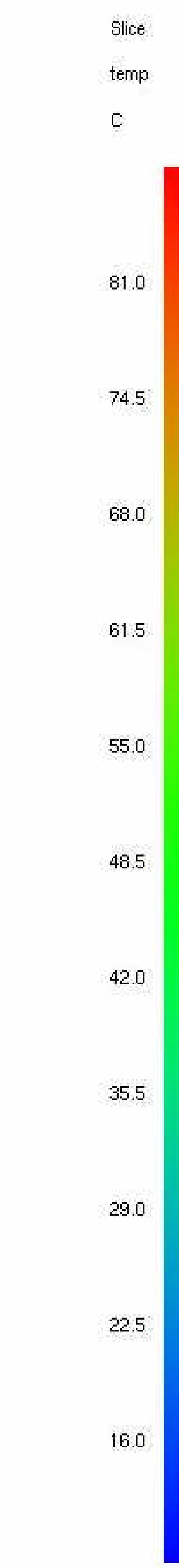


Figure C2.9, Temperature slice at t=600 seconds



C3. Further simulations - Increased reservoir length, 90 meter

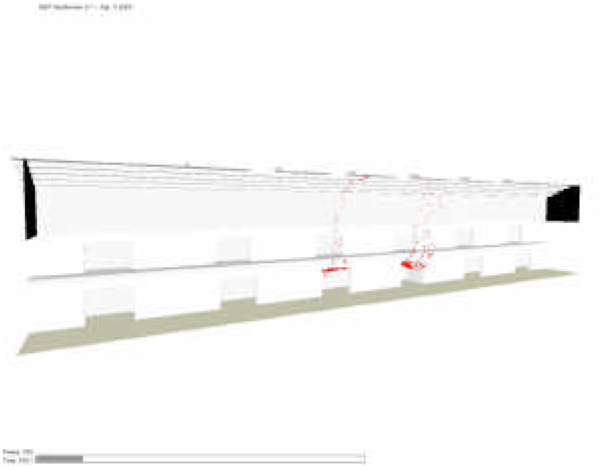


Figure C3.1, Particulate tracings at t=100 seconds

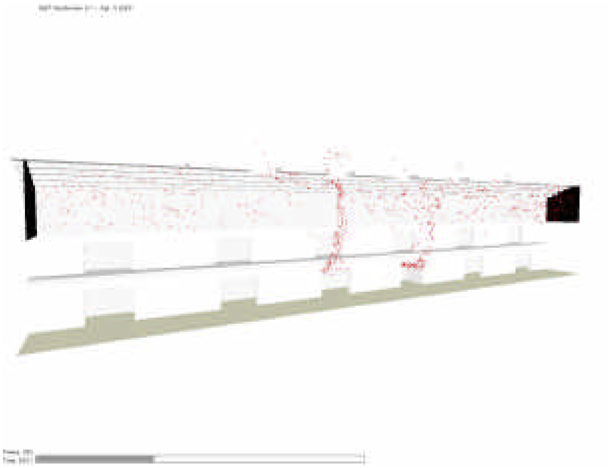


Figure C3.2, Particulate tracings at t=150 seconds

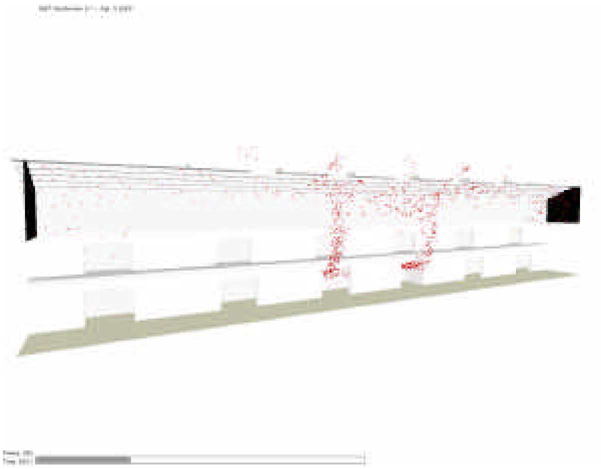


Figure C3.3, Particulate tracings at t=200 seconds

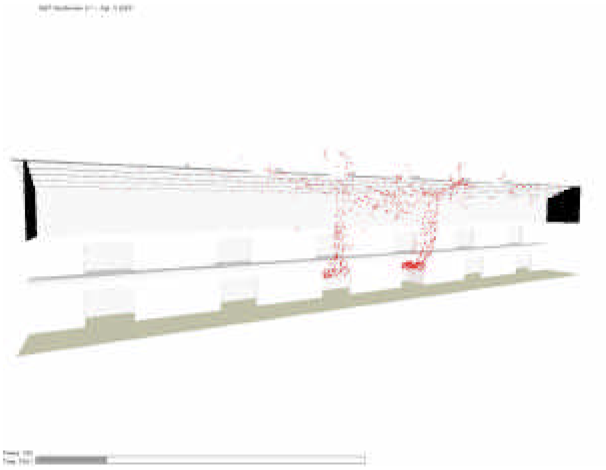


Figure C3.4, Particulate tracings at t=250 seconds

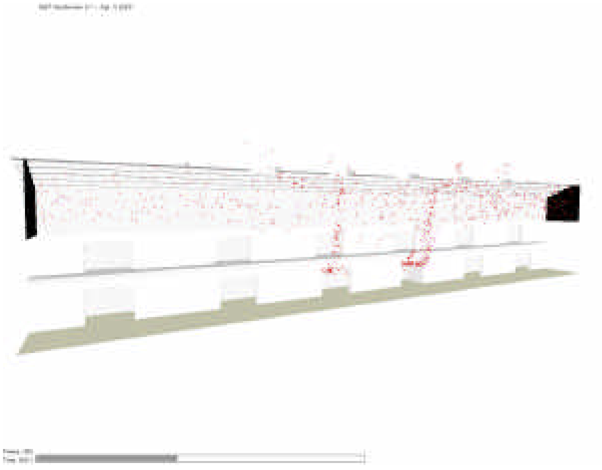


Figure C3.5, Particulate tracings at t=300 seconds

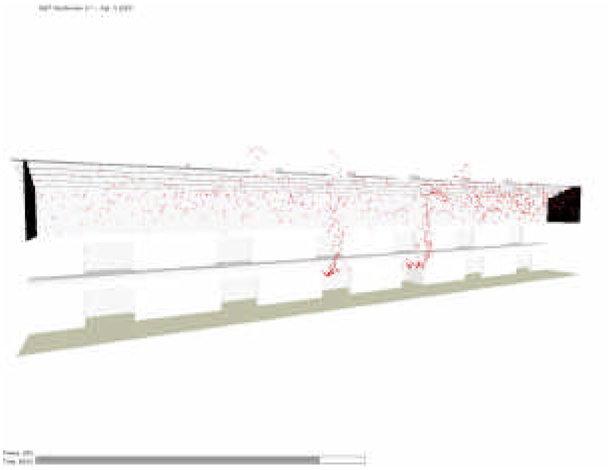


Figure C3.6, Particulate tracings at t=600 seconds

Temperature

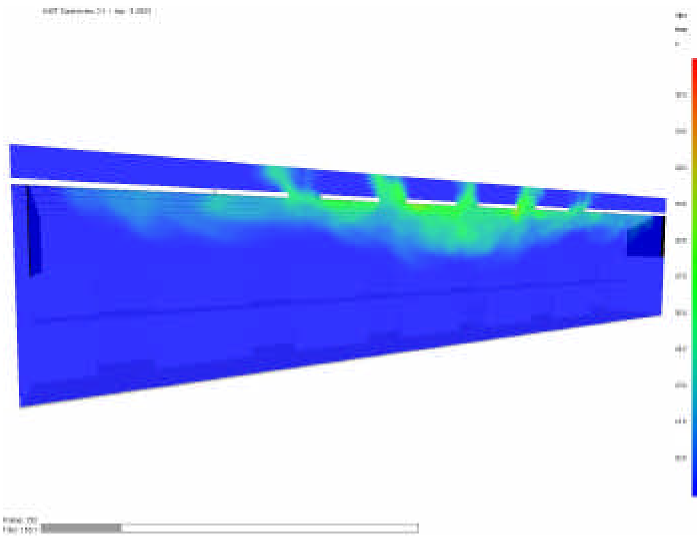


Figure C3.7, Temperature slice at t=150 seconds

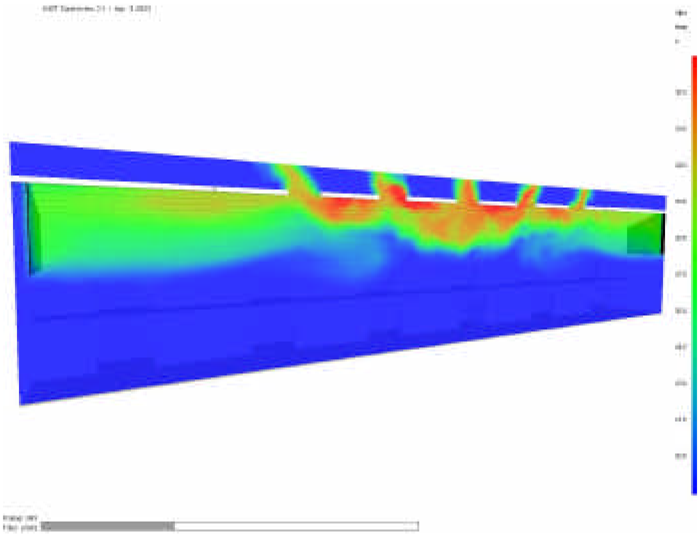


Figure C3.8, Temperature slice at t=250 seconds

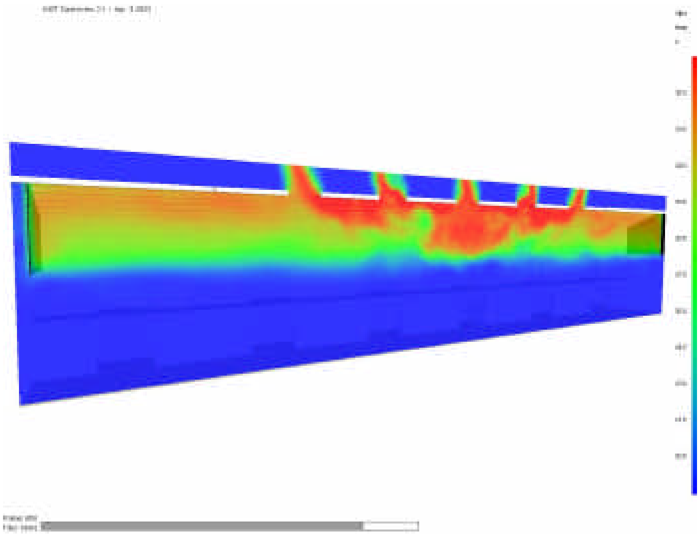
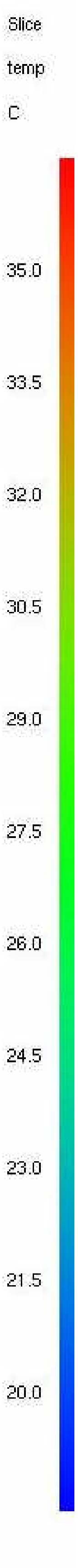


Figure C3.9, Temperature slice at t=600 seconds



C4. 180 meter long smoke reservoir

Particles

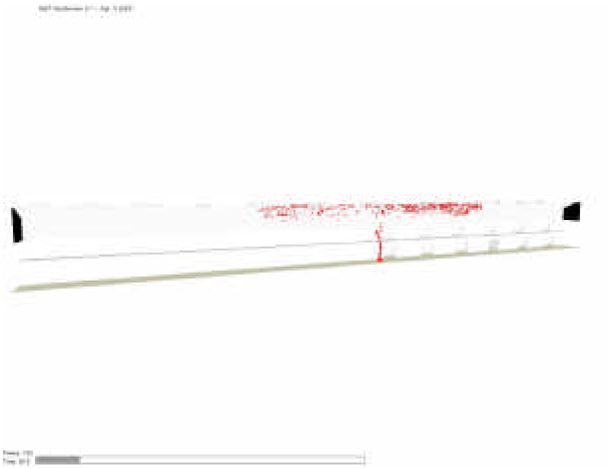


Figure C4.1, Particulate tracings at t=100 seconds

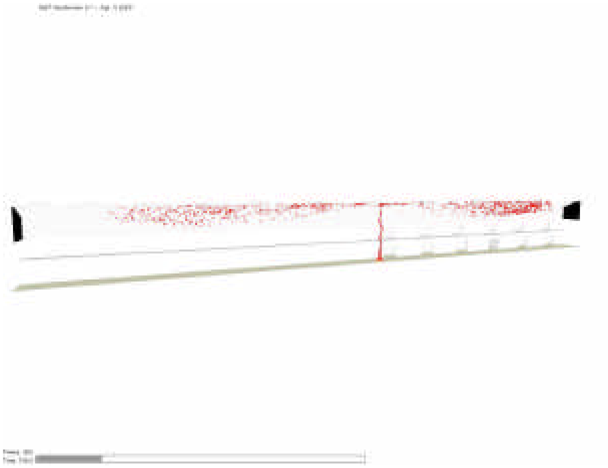


Figure C4.2, Particulate tracings at t=150 seconds

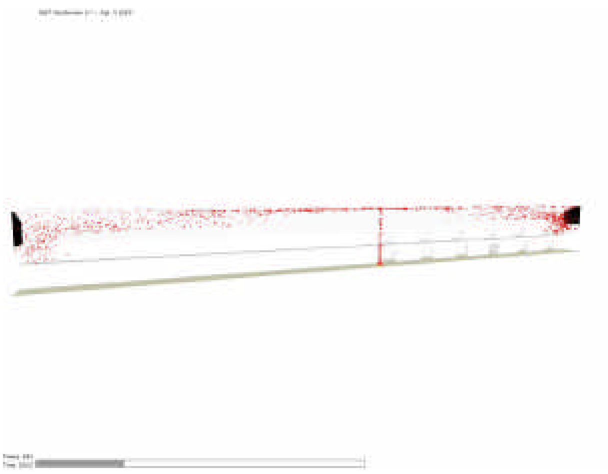


Figure C4.3, Particulate tracings at t=200 seconds

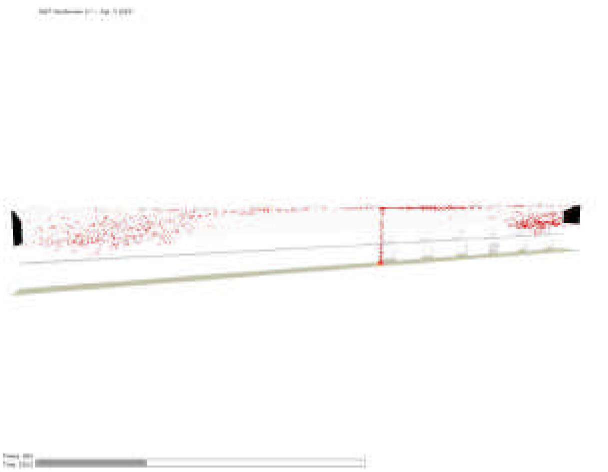


Figure C4.4, Particulate tracings at t=250 seconds

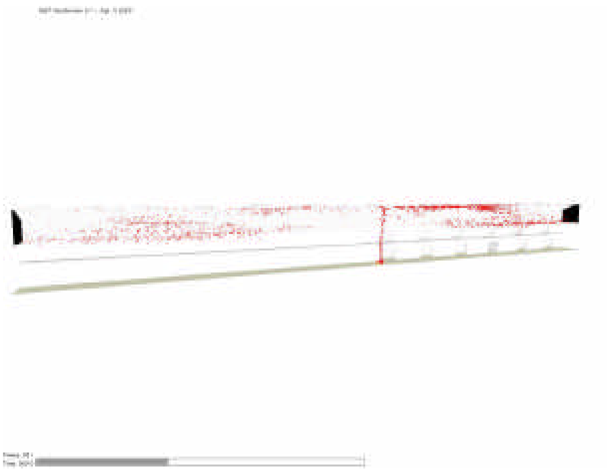


Figure C4.5, Particulate tracings at t=300 seconds

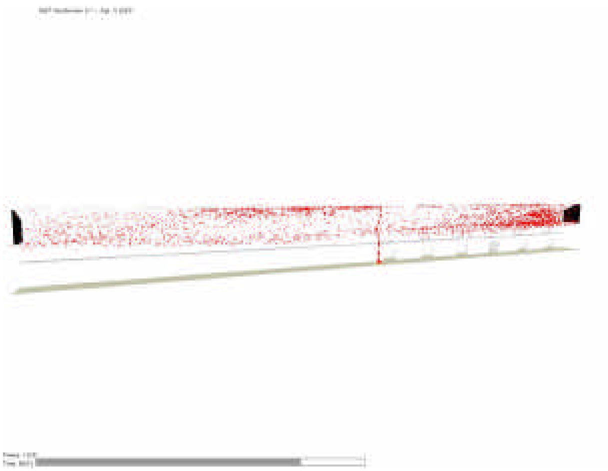


Figure C4.6, Particulate tracings at t=600 seconds

Temperature

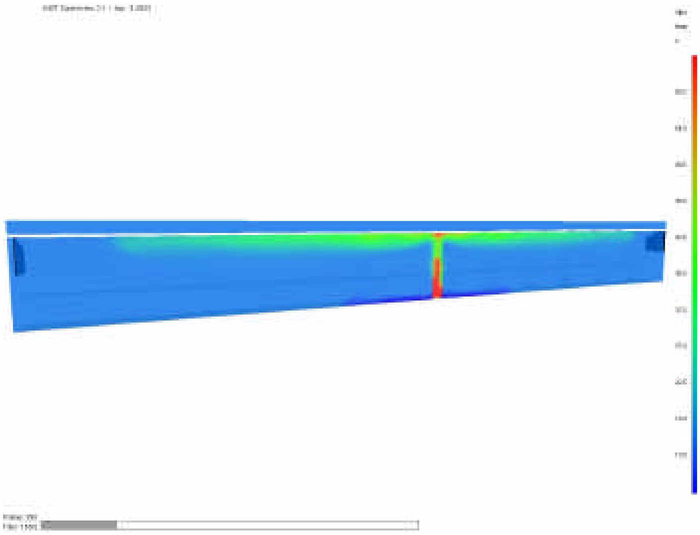


Figure C4.7, Temperature slice at t=150 seconds

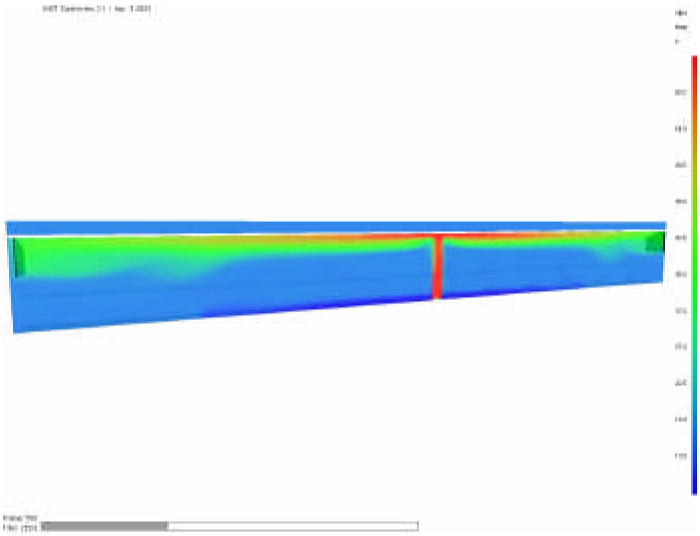


Figure C4.8, Temperature slice at t=250 seconds

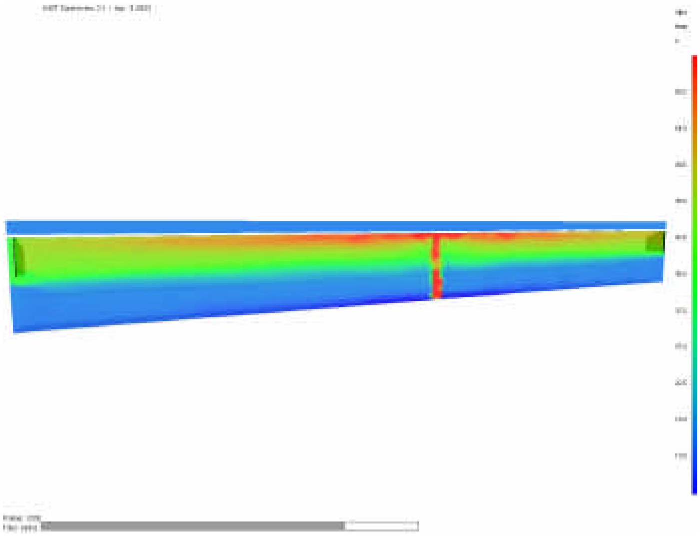


Figure C4.9, Temperature slice at t=600 seconds

C5. Temperature differences within the atrium

Particles

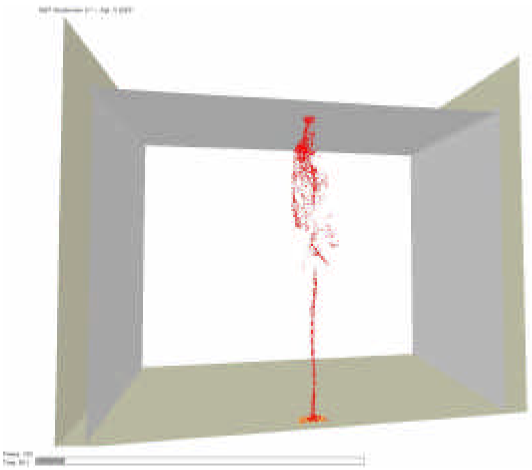


Figure C5.1, Particulate tracings at t=50 seconds

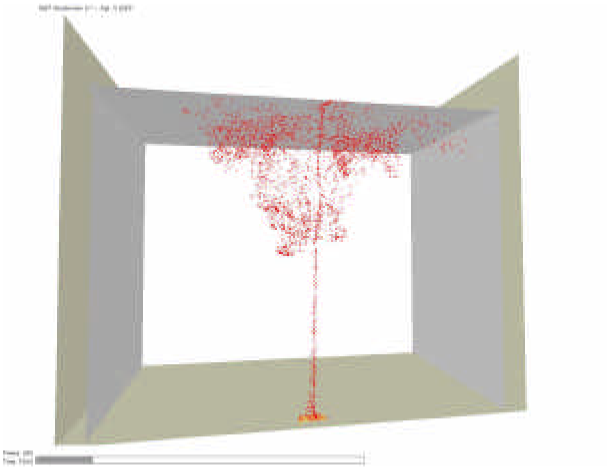


Figure C5.2, Particulate tracings at t=100 seconds

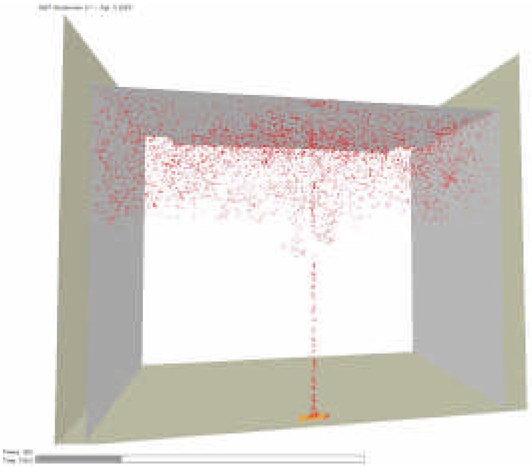


Figure C5.3, Particulate tracings at t=150 seconds

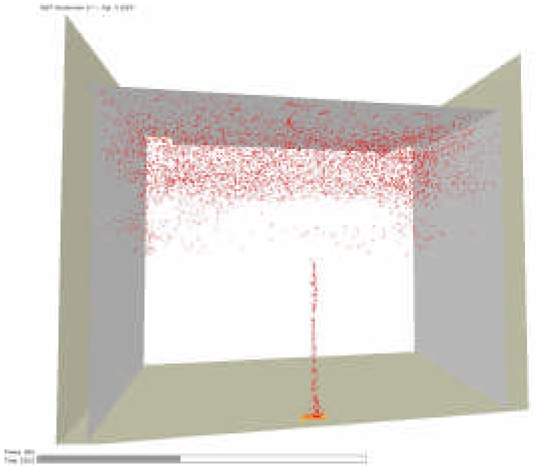


Figure C5.4, Particulate tracings at t=250 seconds

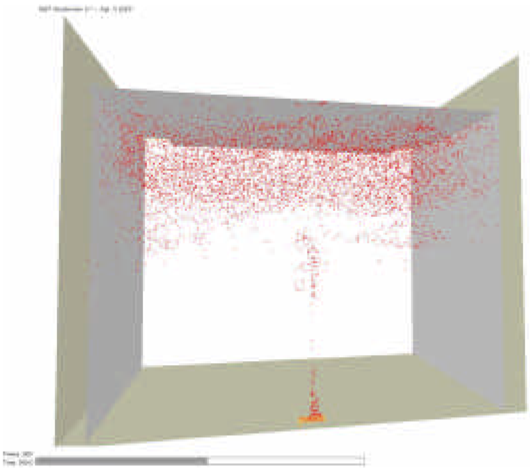


Figure C5.5, Particulate tracings at t=300 seconds

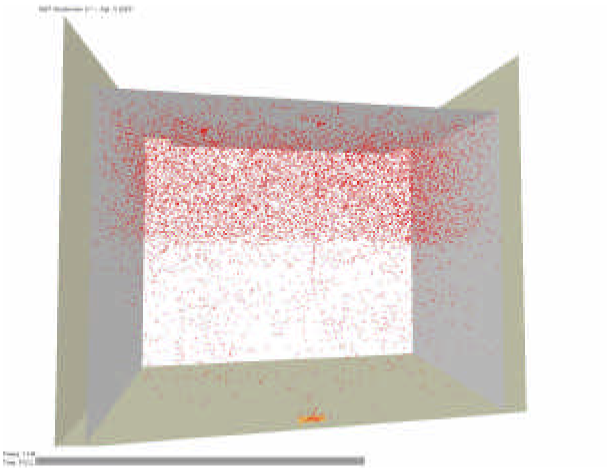


Figure C5.6, Particulate tracings at t=600 seconds

Temperature

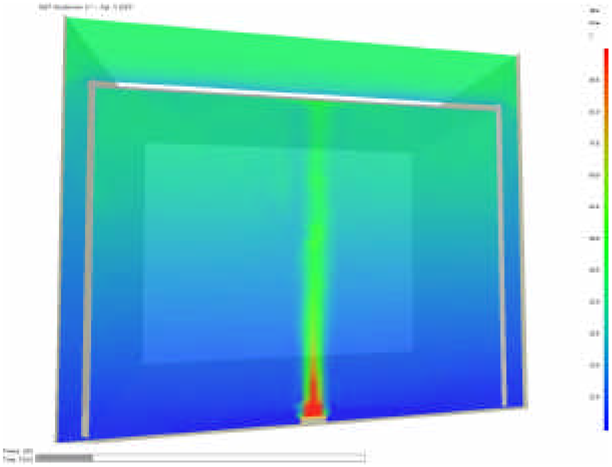


Figure C5.7, Temperature slice at t=100 seconds

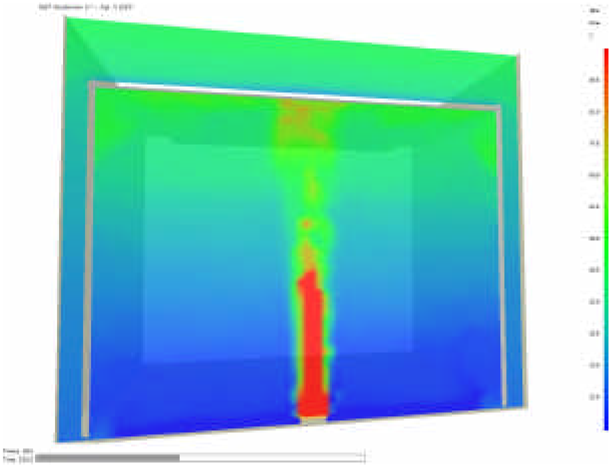


Figure C5.8, Temperature slice at t=250 seconds

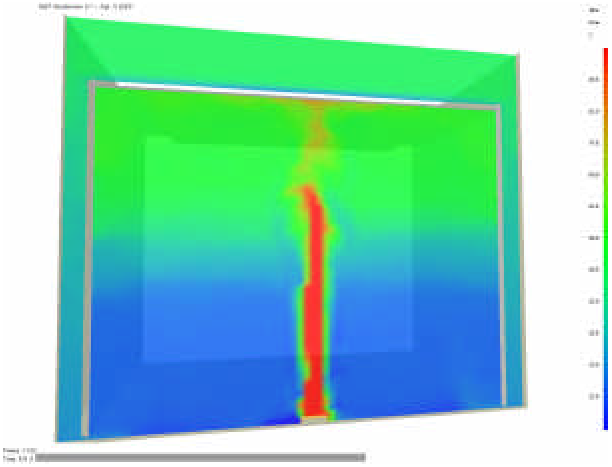


Figure C5.9, Temperature slice at t=600 seconds

C6. Heat loss to a cold glass ceiling

Particles

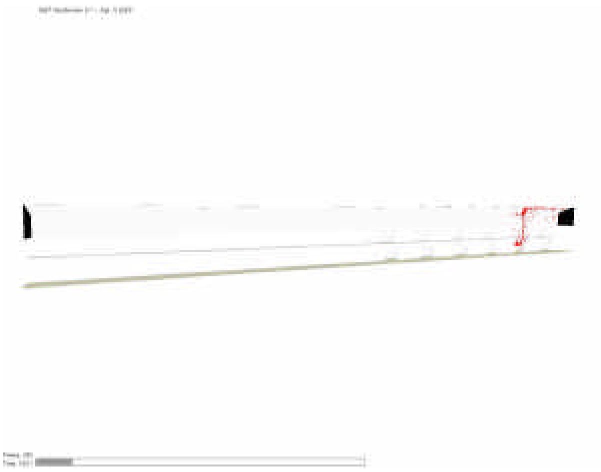


Figure C6.1, Particulate tracings at t=100 seconds

Figure C6.2, Particulate tracings at t=150 seconds

Figure C6.3, Particulate tracings at t=200 seconds

Figure C6.4, Particulate tracings at t=250 seconds

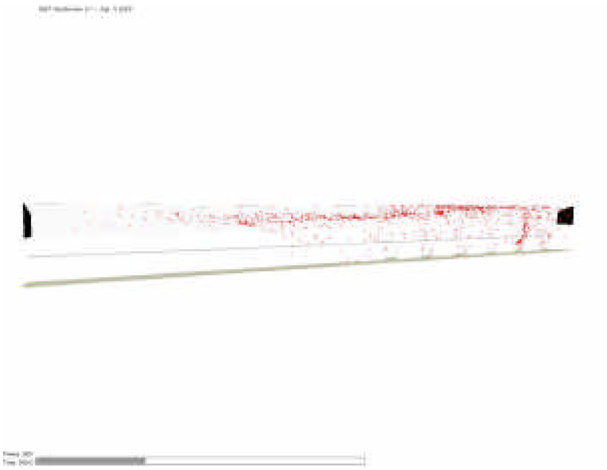


Figure C6.5, Particulate tracings at t=300 seconds

Figure C6.6, Particulate tracings at t=600 seconds

Temperature

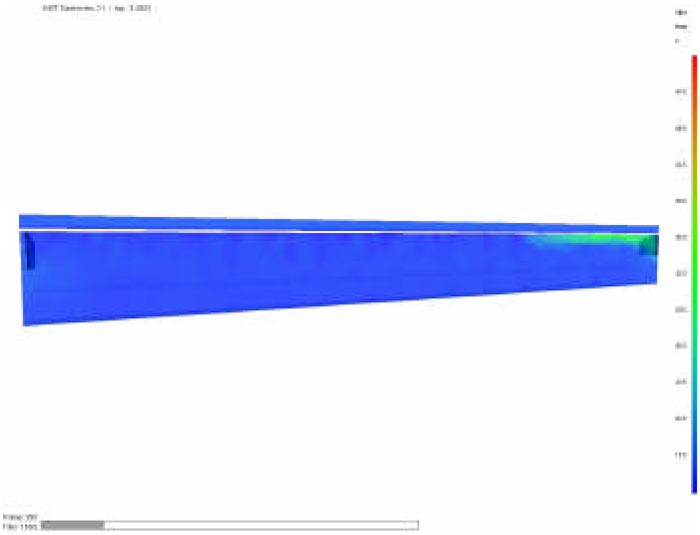


Figure C6.7, Temperature slice at t=150 seconds

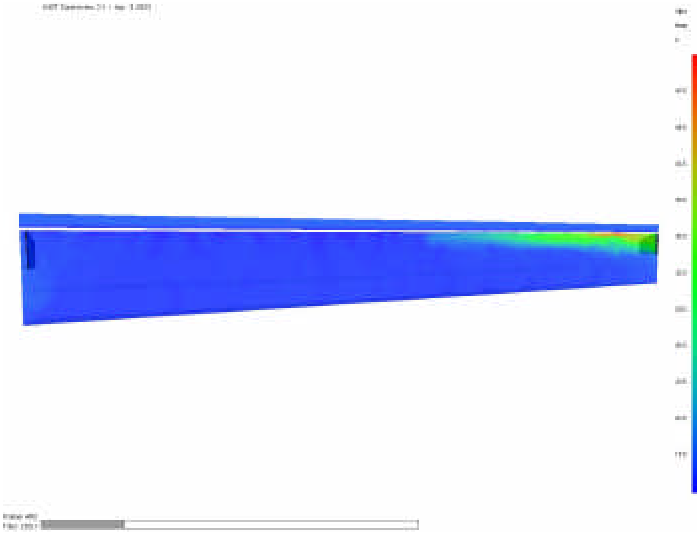


Figure C6.8, Temperature slice at t=250 seconds

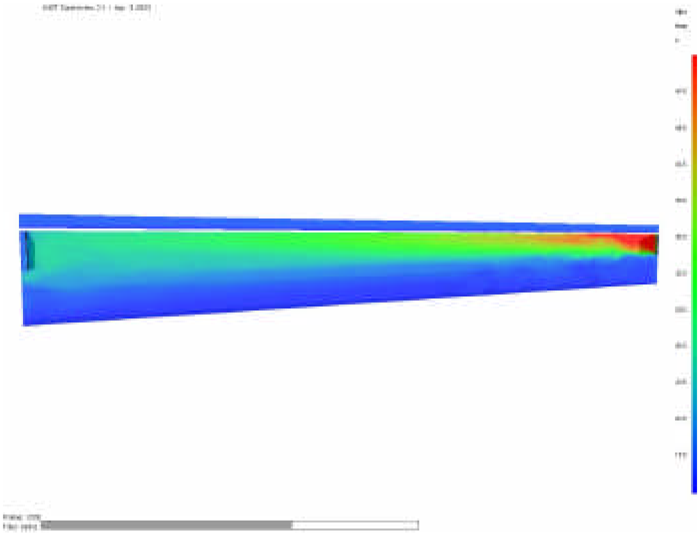
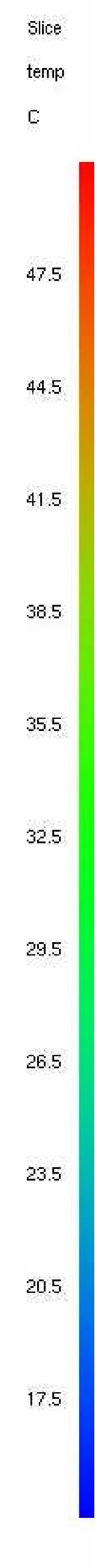


Figure C6.9, Temperature slice at t=600 seconds



C7. Increased cell resolution

Particles

Figure C7.1, Particulate tracings at t=100 seconds

Figure C7.2, Particulate tracings at t=150 seconds

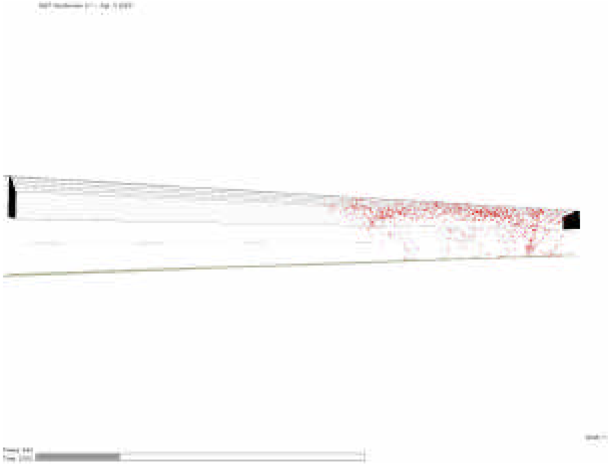


Figure C7.3, Particulate tracings at t=230 seconds

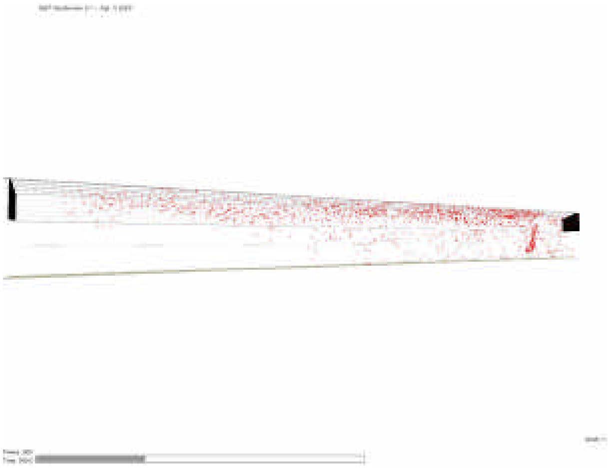


Figure C7.4, Particulate tracings at t=300 seconds

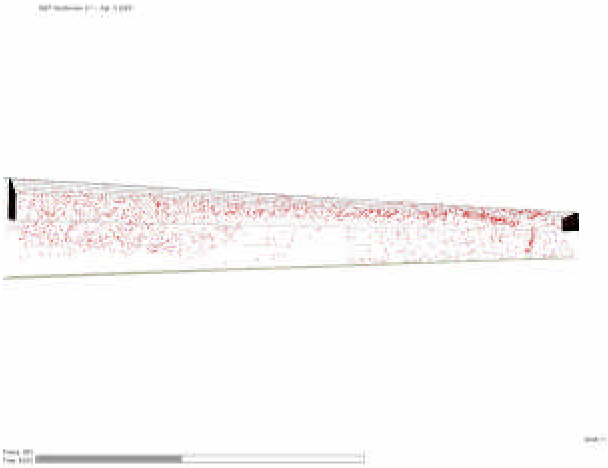


Figure C7.5, Particulate tracings at t=400 seconds

Figure C7.6, Particulate tracings at t=600 seconds

Temperature

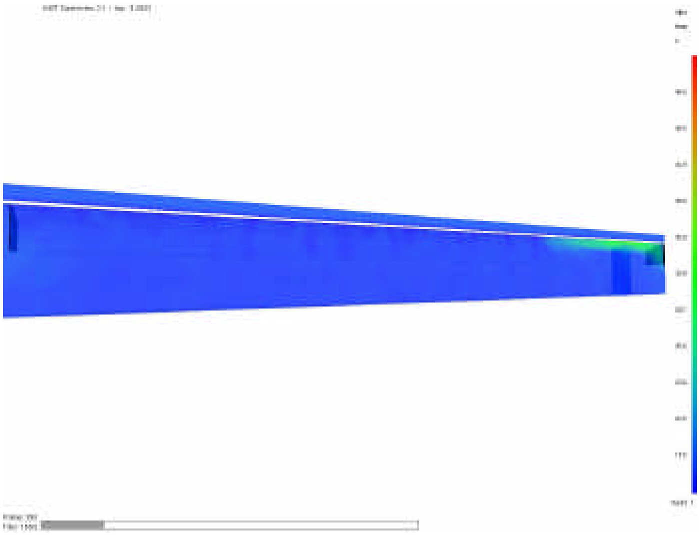


Figure C6.7, Temperature slice at t=150 seconds

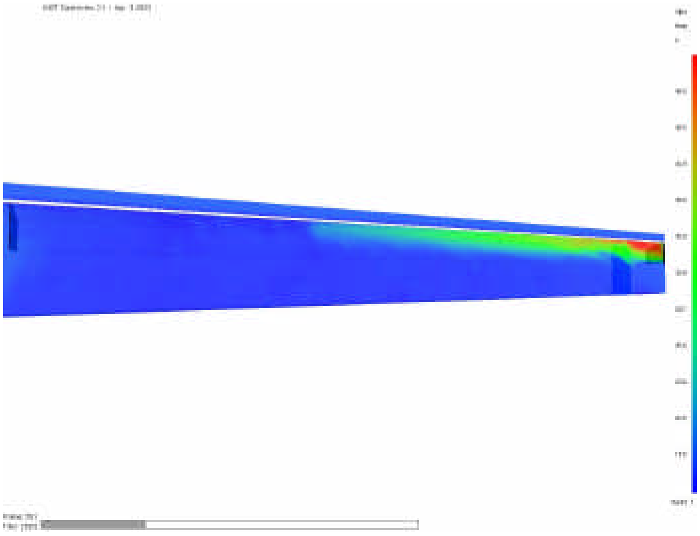


Figure C6.8, Temperature slice at t=250 seconds

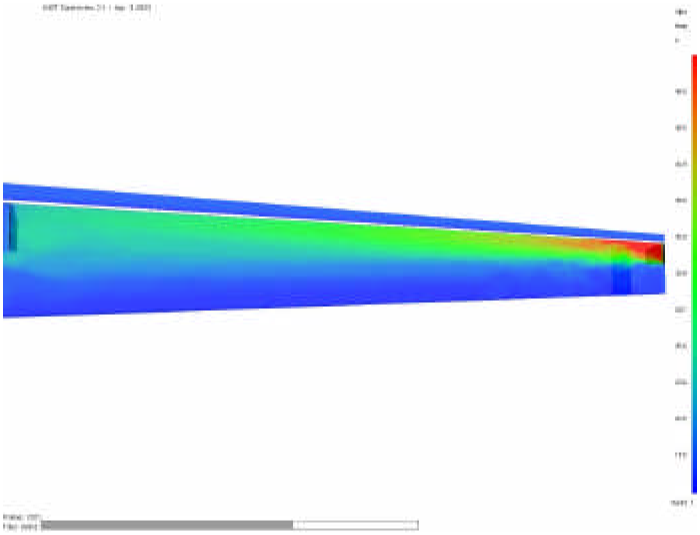


Figure C6.9, Temperature slice at t=600 seconds