MASTER

Smoke movement in fire situations
CFD-utilization in car park Fleerde

Hegeman, S.T.G.D.

Award date:
2009

Link to publication
Smoke movement in fire situations

*CFD-utilization in car park Fleerde*

By:
S.T.G.D. Hegeman
December 2008

Under supervision of:
Prof. Dr. Ir. J.Hensen (TU/e)
Dr. Ir. M.G.L.C Loomans (TU/e)
Ir. A.D. Lemaire (Efectis)
Ir. L.M. Noordijk (Efectis)
Due to growing population and level of prosperity, spatial planning in the Netherlands more and more accounts for the necessity of efficient building on construction sites in order to minimize the weight on limited available natural environment. High rise buildings as well as constructions beneath surface level pop-up in both urban and rural environments, offering an answer to these developments. The goal, accommodation of larger amounts of people and stock, is reached, but additional efforts have to be made in order to equal the level of functionality, convenience and/or safety. Since Dutch building regulations are limited suitable for both high-rise as well as beneath surface-level buildings, special attention has to be given to these types of structures. E.g. fire safety becomes more and more critical in case of extended egress routes and limited natural ventilation, both occurring in these non-common buildings. Proper predictions of heat and smoke movement and fire spread enable us to design a safe environment for work, living and recreation in these building types.

More and more, heat and smoke movement is investigated using CFD-simulations. Results are used to prove a safety level equal to the level required according to building regulations (which is a legal procedure by using the equivalence principle). Human safety (by providing smoke free egress zones), building conservation (by providing limited temperatures) and safe fire suppression by the fire department (by providing sight, access and appropriate conditions) can be obtained or supported by the use of proper ventilation conditions. The working of the ventilation system is influenced by capacity, enclosure geometry as well as dimension and location of system components. Although some general design rules are available, detailed calculations are necessary for thorough appraisal of the system design.

In this project the focus is laid on smoke movement in enclosed car parks. Jet fan ventilation, commonly used in tunnel safety solutions, is utilized to provide a smoke free zone upstream of the fire location. The technique is used more and more in both existing as well as new car parks and although the proper working of these systems can, within limitations, be demonstrated using CFD, several aspect still are subject of discussion. Goal of this project is to provide some insight in the problems one faces using CFD-simulations, also used in lots of other research field (figure 1), for the above described purpose. The CFD-code itself, or the governing equations for fluid flow, will not be discussed. On the contrary, the level of detail to which jet fans, obstacles and fire properties are introduced is discoursed and examined in more detail.

Figure 1: CFD: Computational Fluid Dynamics

---

the science of determining a numerical solution to the governing equations of fluid flow whilst advancing the solution through space or time to obtain a numerical description of the complete flow field of interest [NASA].

Studies of aerodynamics form a common used example in relation to CFD-simulations.
Contents

Foreword .................................................................................................................. 1
Contents .................................................................................................................. 3
Figures ................................................................................................................... 5
Tables ..................................................................................................................... 6
Report summary ..................................................................................................... 7
Samenvatting van de rapportage (report summary in Dutch) ................................ 9
Preface ................................................................................................................... 11
Acknowledgements ............................................................................................... 12
Introduction ......................................................................................................... 13

Part 1

1. Introduction ........................................................................................................ 15
2. Practical implications; the use of CFD and its anchoring in building regulations .. 18
3. Properties of fluid, fluid mechanics in fire situations ....................................... 19
4. Utilization of CFD for the prediction of fluid behaviour in fire situations .......... 21
    4.1 Modelling of fluid flow .................................................................................. 22
        4.1.1 Zone models (in fire situations) ............................................................ 22
        4.1.2 Field models ........................................................................................ 23
    4.2 Field modelling: turbulent flow behaviour .................................................. 25
        4.2.1 General modelling approaches for turbulence ..................................... 25
        4.2.2 Turbulence in fire situations ................................................................. 26
    4.3 Field modelling: use of radiation models .................................................... 27
        4.3.1 General modelling approaches for radiation ........................................ 27
        4.3.2 Radiation in fire situations ................................................................. 28
    4.4 Field modelling: inclusion of combustion phenomena .................................. 28
    4.5 Field modelling: inclusion of smoke properties in relation to sight ............... 29
5. Proper use of CFD, limitations and validation .................................................. 32

Part 2

1. Introduction ........................................................................................................ 34
2. Description of the 2D room-fire case ................................................................ 35
    2.1 Description of the original full-scale test facility ......................................... 35
    2.2 Geometry and modelling .............................................................................. 36
3. Results ............................................................................................................... 38
    3.1 Development of temperature and velocity .................................................... 38
4. Discussion .......................................................................................................... 41
    4.1 Balance and convergence .............................................................................. 41
5. Study of variants ............................................................................................... 43
    5.1 Mesh sensitivity ........................................................................................... 43
    5.2 Increased amounts of energy in the domain ................................................ 47
        5.2.1 Influence of radiation ............................................................................ 47
        5.2.1a -10% radiative heat loss ...................................................................... 47
        5.2.1b +10% radiative heat loss ..................................................................... 48
        5.2.2 Influence of the environmental temperature ........................................ 49
    5.3 Change in geometrical characteristics and properties ................................. 50
        5.3.1 Constant wall temperature .................................................................... 50
        5.3.2 Non-symmetrical walls alongside ......................................................... 51
        5.3.3 Alternative flame geometry .................................................................. 53
6. Overall conclusions and discussion .................................................................. 54
    6.1 Original case and mesh study ....................................................................... 54
    6.2 Increased temperature as a result of increased amounts of energy in the domain .... 54
    6.3 Altered temperature and velocity profiles due to change in geometrical characteristics and properties 55

Part 3

1. Introduction ........................................................................................................ 57
2. Description of the Fleerde-case ....................................................................... 58
    2.1 Description of the original full-scale test facility ......................................... 58
    2.2 Thrust ventilation in car park Fleerde .......................................................... 59
    2.3 Measurements taken during the full-scale fire tests .................................... 60
        2.3.1 Air temperature ..................................................................................... 60
        2.3.2 Radiation ............................................................................................. 60
        2.3.3 Heat produced by the fire ..................................................................... 61
        2.3.4 Optical density ..................................................................................... 62
    2.3.5 Velocity of air .......................................................................................... 62
3. Modelling of car park Fleerde

3.1 Names and versions of software codes used

3.2 Geometry and mesh generation

3.2.1 Mesh properties

3.3 Problem set up

3.3.1 Rate of heat/smoke release

3.3.2 Air exhaust at the rear wall

3.3.3 Modelling of the jet fans

3.3.4 Gravitational correction

3.3.5 Radiation and wall conditions

3.4 Initialization and solution

3.4.1 Development stages

4. Validation of the simulation results

4.1 Convergence and consistency

4.2 Comparison with previous obtained results (measurement and VESTA-simulations)

4.2.1 Modifications in the model

4.2.2 Remarks

4.2.3 Results

4.2.4 Discussion

4.2.5 Conclusion

4.3 Discussion

4.3.1 Comparison of the results using the contour plots

4.3.2 Comparison of the results using the height based graphs

4.3.3 Comparison of the results using the scatter plots

5. Study of alternatives

5.1 Alternative (coarse) grid

5.1.1 Modifications in the model

5.1.2 Remarks

5.1.3 Results

5.1.4 Discussion

5.1.5 Conclusion

5.2 Alternative flame volume

5.2.1 Modifications in the model

5.2.2 Remarks

5.2.3 Results

5.2.4 Discussion

5.2.5 Conclusion

5.3 Alternative number of cars in the car park

5.3.1 Modifications in the model

5.3.2 Remarks

5.3.3 Results

5.3.4 Discussion

5.3.5 Conclusions

5.4 Alternative modelling of the jet fans

5.4.1 Modifications in the model

5.4.2 Remarks

5.4.3 Results

5.4.4 Discussion

5.4.5 Conclusion

5.5 No gravitational correction

5.5.1 Modifications in the model

5.5.2 Remarks

5.5.3 Comparison to results of previous simulations by TNO-CvB

5.5.4 Comparison to results from the basic case

5.5.6 Conclusions

5.6 No structural beams in the car park

5.6.1 Modifications in the model

5.6.2 Remarks

5.6.3 Results

5.6.4 Discussion

5.6.5 Conclusion

6. Overall conclusions and discussion

108
Figures

Figure 1: CFD: Computational Fluid Dynamics .................................................. 2
Figure 2: Decreasing walking speed in relation to smoke compounds and thickness ........ 16
Figure 3: Evacuation modelling ........................................................................... 16
Figure 4: Stack effect; ......................................................................................... 19
Figure 5: Fire triangle ......................................................................................... 20
Figure 6: Zone modelling .................................................................................... 22
Figure 7: Turbulence and smoke ........................................................................ 25
Figure 8: Visualization of the shopping mall, Fleerde, Heselden .................................. 35
Figure 9: Simplified representation of the shopping-mall test facility [meters] .................. 36
Figure 10: Measurement positions (Heselden) ..................................................... 37
Figure 11: Mesh Roomfire case at z=3 .................................................................. 37
Figure 12: Temperature (lines, legend (°C)) and velocity (vectors); original case ............ 38
Figure 13-a-f: Comparison of original results with previous simulation results and measurement data .................................................. 39
Figure 14 a-f: Temperature development during the first minute; original case .............. 40
Figure 15 a-f: Comparison of measurement data with simulation results and other data .......................... 43
Figure 16: Mesh study, denser mesh .................................................................... 43
Figure 17: Mesh study, coarser mesh .................................................................... 43
Figure 18 a-f: Comparison densified mesh with original mesh ............................... 45
Figure 19 a-f: Comparison coarsened mesh with original mesh .............................. 46
Figure 20a-b: Comparison of original results with previous simulation results and measurement data .......................................................... 47
Figure 21a-b: Comparison of increased convective heat transfer with original results and other data ........................................................................ 47
Figure 22a-b: Comparison of increased environmental temperature with original results and other data ................................................................. 49
Figure 23a-b: Comparison of constant temperature walls with original results and other data ................................................................. 50
Figure 25a-b: Comparison of wall-as-wall with original results and other data; centre-line of the room (z=3) ................................................................. 51
Figure 26: Walls alongside as wall (alternative case), temperature (legend (°C)) at centre-line of the room ................................................................. 52
Figure 27: Perspective view of transverse temperatures (legend (°C)) ; ...................... 52
Figure 28: Perspective view of longitudinal temperatures (legend (°C)) ...................... 52
Figure 29: Geometry with alternative flame dimensions ........................................ 53
Figure 30: Temperature (legend (°C)) using altered dimensions for the flame geometry (z=3) ................................................................. 53
Figure 31a-b: Comparison of alternative flame dimensions with the original case (z=3) ..................................................................................... 53
Figure 32: Photos taken during the full-scale tests at car park Fleerde ....................... 58
Figure 33: Floor plan of car park Fleerde ............................................................. 58
Figure 34: The principle of thrust ventilation in fire situations in car parks ................. 59
Figure 35: Placement of thermocouples in car park Fleerde, top view ..................... 60
Figure 36: Rate of heat release ............................................................................ 61
Figure 37: Visualisation of smoke release ................................................................ 71
Figure 38: Mesh generated for car park Fleerde – basic case .................................. 65
Figure 39: Flow rate (exhaust opening) and heat release in UDF .............................. 67
Figure 40: Fixed velocities over the height of the velocity fields ............................. 69
Figure 41: Visualization of the fixed velocity correction .......................................... 70
Figure 42: Visualization of the fixed velocity, enlarged; vectors are scaled ............... 70
Figure 43: Longitudinal intersection over jet fan and correction ............................. 70
Figure 44: Visualization of the fixed velocity at z=21,4; vectors are scaled ............... 71
Figure 45: Principle of the gravitational correction in substitution for modelling of the slope .................................................................................................. 71
Figure 46: Slope in the car park, approximately 3.4° incline (scaled figure) ............... 71
Figure 47: Temperature in fire plume, without gravitational correction .................... 72
Figure 48: Velocity magnitude in fire plume, without gravitational correction .......... 72
Figure 49: Temperature in fire plume, with gravitational correction .......................... 72
Figure 50: Velocity magnitude in fire plume, with gravitational correction ................ 72
Figure 51: Alternative flame volume (Y-view) ...................................................... 72
Figure 52: Identification of the planes alongside the car park .................................. 72
Figure 53: Presentation of the simulation results in contour and vector plots .............. 79
Figure 54: Scatter plot ......................................................................................... 80
Figure 55: Data points for mutual comparison ..................................................... 80
Figure 56: Comparison between measurement data and simulation results, odd numbered points ................................................................. 83
Figure 57: Comparison between measurement data and simulation results, even numbered points ............................................................................... 84
Figure 58: Scatter plot for comparison of measurement data with simulation results .... 85
Figure 59: Mesh generated for car park Fleerde – alternative (coarse) mesh .............. 90
Figure 60: Influence of mesh size at temperatures and x-velocity, data points 10 – 15 ................................................................. 91
Figure 61: Influence of mesh size at temperatures and x-velocity, data points 10 – 15 ................................................................. 91
Figure 62: alternative flame volume (Y-view) ...................................................... 92
Figure 63: Comparison temperatures basic case and temperatures alternative flame volume ................................................................. 93
Figure 64: Comparison slight length basic case and slight length alternative flame volume ................................................................. 93
Figure 65: Comparison velocity basic case and velocity alternative flame volume .......... 93
Figure 66: Flame volume - sectional view (z=19,1) for optical density at t=420, t=600 and t=840 s ................................................................. 94
Figure 67: Comparison temperatures basic case and temperatures when no cars are present .......................... 96
Tables

Table 1: overview of tolerated fire exposure conditions ................................................................. 18
Table 2: Set of k-ε model constants (Launder and Spalding [3]) .......................................................... 26
Table 3: overview of experimental data versus results from simulations ........................................ 32
Table 4: fluxes through boundaries ................................................................................................. 50
This report describes the result of the graduation project of Saskia Hegeman, at the Technical University of Eindhoven. Main aspect in the project is the heat and smoke movement in car park fires, supported by a jet fan ventilation system.

Apart from the introduction and theoretical considerations, the project investigates two situations: the Roomfire case, and the Fleerde case.

Roomfire case
Goal of the Roomfire case is to reproduce measurement as well as previously obtained simulation results using the CFD-code FLUENT. In addition, the case provides the opportunity to do a short study of variants. The case describes a room (about 160 m²) with a full width opening at one short side of the room and the outside environment partly introduced in the modelling. A volumetric heat source near the back of the room represents the fire.

Results of the FLUENT simulation show relatively good agreement with results obtained by Markatos in a former simulation. A 20-40 K temperature underestimation in the upper layer, in comparison to the measurement results, is found.

As one of the first of the variant studies shows, this is probably due to overestimation of the radiative heat transfer. The radiation is taken into account by decreasing the total heat release by the fire, which is only a rough estimation. In the Fleerde case, a more sophisticated radiation model will be used. In all variants, except for the case in which the flame geometry is altered, the typical separation between the upper and lower layer is found. The change of wall properties, in the new situation responsible for a +12.1% heat release and decreased ventilation, results in a higher upper layer temperature.

Fleerde Case
The Fleerde Case is the main subject in the graduation project and describes a 85x33m car park in which a fire develops in one of the cars. In 1999 full scale fire tests were carried out to prove the applicability of jet fan smoke ventilation in car parks.

In the Fleerde case, the first goal is to reproduce the results obtained from the full-scale fire test performed by TNO in the car park, using the CFD-code FLUENT. Second goal is to qualify the influence of some non-design specific variables on the heat and smoke movement pattern; therefore, three objectives can be distinguished:
1. Reproduction of the heat and smoke movement in car park Fleerde and comparison of the results from the CFD-simulation(s) with measurement data obtained by TNO;
2. Qualification of the influence of several non-design specific modelling parameters (e.g. the number of cars present in the car park) on the heat and smoke movement pattern;
3. Qualification of the influence of several design parameters (e.g. structural beams or flat ceiling) on the heat and smoke movement pattern;

In addition to geometry and general numerical set-up a number of User Defined Functions is included. These UDF's control heat and smoke production, ventilation conditions and various submodels replacing the slope and refining the jet stream.

The results are compared to the measurement data and despite some relatively small differences, the data points downstream of the fire show relatively good agreement. Instead, in contradiction to the actual situation during the full scale fire test, in the simulation the jet fans were not able to control the smoke movement and create an upstream smoke-free zone.

In addition, wall properties (introduced using an UDF) do not fully describe the actual situation, resulting in near-ceiling temperatures that likely are too low.
Comparison with the VESTA results is based on the FLUENT results in which the gravitational correction for the slope was not taken into account. Omitting the slope was one of the simplifications in the VESTA model. Results showed relative good agreement, both upstream and downstream of the fire location.

Results from the study of variants demonstrate that some assumptions do affect the results, for the better or the worse. Results in which heat and smoke are removed from the car park less efficient, like when cars or structural beams are present in the car park, could result in higher risks in an actual fire situation, especially for fire fighters. These aspects should be subject to further discussion and more profound research.

As expected, the detail to which the jet fans are modelled influences the results: a detailed submodel should be incorporated in simulations and more research is needed to improve the performance of the submodel in use. Also the implementation of the wall model should be reviewed.

Some other aspects, like the size of the volumetric heat source, do not seem to seriously affect the results, but as for all the variants, this is a first result and further examination is necessary in order to be able to draw more thorough conclusions.

Apart from the specific conclusion in relation to both cases, some more general conclusions concerning the use of CFD (in fire safety engineering) can be drawn. At first a CFD model is, as all models are, sensitive to the quality of the input. Knowledge about modeling, thermodynamics and some insight in the impact of simplifications is necessary for proper use, analysis and evaluation of all CFD models.

Second, and more specific for fire safety engineering, the validation of CFD models is thwarted by the lack of proper measurement results, which is a result of difficulties experienced when performing measurements in fire situations. Fortunately, the CFD codes itself have evolved in such a way that, with respect to fire safety engineering, the errors due to uncertainties within the codes itself are in most circumstances subordinate to errors due to inapt use or limited validation options.
Samenvatting van de rapportage (report summary in Dutch)

Dit rapport beschrijft de resultaten van het afstudeerproject van Saskia Hegeman, aan de Technische Universiteit van Eindhoven. Belangrijkste onderwerp in het project is de verplaatsing van warmte en rook, ondersteund door stuwkrachtventilatie, bij een brand in een parkeergarage.

Naast een introductie en een overzicht van de belangrijkste theoretisch achtergronden, kent het project twee hoofdonderwerpen: de Roomfire-studie en de Fleerde-studie.

Roomfire studie
Doel van de Roomfire-studie is om met gebruik van FLUENT, de meetresultaten en resultaten van eerdere simulaties te reproduceren. Daarnaast biedt deze casus de mogelijkheid om een korte variantenstudie uit te voeren. Het betreft een vertrek (ca. 160 m^3) met aan een van de korte zijden een opening over de volledige breedte; een gedeelte van de omgeving is meegenomen in het model. Een volumetrische warmtebron representeert de vlammen in de achterzijde van het vertrek.

De resultaten van de simulatie met FLUENT komen redelijk goed overeen met de resultaten die Markatos in een eerdere simulatie heeft verkregen. In vergelijking tot de meetresultaten wordt de temperatuur in de bovenste laag (de rooklaag) onderschat met ongeveer 20 tot 40 Kelvin. Mogelijk is dit te wijten aan een overschatting van de hoeveelheid warmte die in de vorm van straling aan het vertrek wordt afgegeven. Deze hoeveelheid straling wordt geïntroduceerd in het model door de totale warmteafgifte door de vlammen te verlagen, maar deze benadering geeft een zeer grove schatting. In de Fleerde-studie wordt een meer uitgebreid stralingsmodel gebruikt.

Behalve voor de variant waarin de vlamgeometrie wordt aangepast, blijkt in alle varianten de typische scheiding tussen een onderste en bovenste luchtlaag te ontstaan. Het veranderen van de wandeigenschappen, welke in de nieuwe situatie verantwoordelijk zijn voor een extra warmteoverdracht van 12,1% terwijl de ventilatie in het vertrek afneemt, resulteert in een hogere temperatuur in de rooklaag.

Fleerde studie
De casus Fleerde is het belangrijkste onderwerp in het afstudeerproject en beschrijft een parkeergarage (85x33 meter) waarin in een van de auto's een brand ontstaat. In 1999 zijn full-scale brandproeven uitgevoerd om de toepasbaarheid van stuwdrukventilatie in parkeergarages te beoordelen.

Het eerste doel van de Fleerde-studie is opnieuw om meetresultaten uit een full-scale brandproef, uitgevoerd door TNO in de parkeergarage, te reproduceren met behulp van de CFD-code FLUENT. Het tweede doel is om de invloed van een aantal niet-ontwerpspecifieke variabelen op de warmte- en rookverspreiding in de parkeergarage te kwalificeren. Drie doelstellingen kunnen worden onderscheiden:
1. Reproductie van de warmte- en rookverspreiding in parkeergarage Fleerde met behulp van CFD-simulaties en een vergelijking van de resultaten met meetdata van TNO;
2. Bepaling van de invloed van een aantal niet-ontwerpspecifieke parameters (zoals het aantal auto's in de garage) op het gedrag van de warmte en rook in de garage;
3. Bepaling van de invloed van een aantal algemene ontwerpeuzeuges (zoals het al dan niet toepassen van een liggerstructuur onder het plafondvlak) op het gedrag van de warmte en rook in de garage.

Naast het model van de geometrie en de reguliere instellingen in FLUENT is een aantal extra zogenaamde User Defined Functions toegevoegd aan het model. Deze UDF's
beschrijven warmte- en rookproductie, ventilatie en submodellen ter vervanging van de helling of ter verfijning van de straal uit de stuwdrukvventilator.

De resultaten zijn vergeleken met de meetdata en ondanks enkele (relatief kleine) verschillen laten de monitorpunten stroomafwaarts van de brand behoorlijk goede overeenkomsten zien. De stuwdrukvventilatie bleek echter in de simulatie niet in staat verspreiding van rook en warmte naar de zone stroomopwaarts van het vuur te voorkomen. Dit komt niet overeen met de situatie zoals daadwerkelijk is ontstaan tijdens de brandproeven.

Daarnaast ontstaat het vermoeden dat de UDF die de wandeigenschappen beschrijft niet helemaal juist (geïmplementeerd) is, waardoor dicht bij het plafond temperaturen ontstaan die waarschijnlijk te laag zijn.

Vergelijking met de resultaten uit VESTA is gebaseerd op de resultaten waarin de zwaartekrachtcorrectie niet is meegenomen; een van de vereenvoudigingen in het VESTA-model. Deze resultaten komen redelijk goed met elkaar overeen, zowel stroomop- als stroomafwaarts van de brandlocatie.

Resultaten uit de variantenstudie laten zien dat een aantal aannamen wel degelijk van invloed is op de resultaten, zowel in positieve als negatieve zin. Wanneer rook en warmte minder efficiënt kunnen worden afgevoerd uit de garage, bijvoorbeeld wanneer auto's of liggers in de garage aanwezig zijn, vormt dit een verhoogde risico bij een daadwerkelijke brandsituatie. Deze aspecten zouden dan ook verder onderzocht moeten worden. Andere simulaties laten zien dat een aantal aspecten nauwelijks van invloed is op de resultaten; bijvoorbeeld het aangenomen volume van de volumetrische warmtebron. Zoals verwacht beïnvloed de mate van detail in het submodel voor de stuwdrukvventilator het resultaat uit de simulatie. Een meer gedetailleerd model zou moeten worden toegepast, en er zou meer onderzoek moeten worden verricht naar verbetering van het te gebruiken submodel.

Naast deze specifieke conclusies kunnen meer algemene conclusies over het gebruik van CFD (in fire safety engineering) worden getrokken. Zoals elk model, is een CFD model gevoelig voor de kwaliteit van de invoer. Enige kennis over modellering, thermodynamica en inzicht in de invloed van vereenvoudigingen is nodig om op een juiste manier met CFD te werken en de resultaten te analyseren en evalueren.

Ten tweede, en meer specifiek voor de fire safety engineering, wordt het valideren van het model bemoeilijkd door een gebrek aan goede meetresultaten, onder andere als gevolg van de specifieke problemen die worden ervaren bij het uitvoeren van metingen in brandzituaties. De software zelf is in zoverre doorontwikkeld dat, voor fire safety engineering, fouten door onzekerheden in de broncode zelf in de meeste gevallen ongedragen zijn aan fouten door onkundig gebruik of een beperkt gevalideerd model.
Preface

This report describes the developments during the graduation process of Saskia Hegeman in the period from January 2007 to March 2008. The graduation, at the Technical University of Eindhoven, is the last phase in the master track Building Physics and Systems which is one of the four units of the faculty Architecture, Building and Planning. Since fire safety science is not encountered for in the master track, and only partially overlaps the common matter of study objectives, the fourth short-term project was included in the graduation program. In this preparation for the graduation process; a both theoretical and practical basis for modelling smoke movement using CFD was achieved. This comprised both reading up on (computational) fluid dynamics as well as principles and phenomena in fire spread and smoke movement, followed by more specific articles discussing a combination of both. Also, practical training with the CFD-code FLUENT was part of this project.

Main goal of the project, and accompanying research questions (both theoretical and practical), are introduced in the introduction (page 14).

Part 1
In section 1-3, the physical aspects related to fire spread and smoke movement are discussed, followed by the mathematical representation of these phenomena. An introduction to CFD is given in section 1-4, which also discusses the specific basic assumptions to be made in order to predict fluid dynamics in fire circumstances. Prior to this, in section 1-2, the role of CFD-simulations in Dutch building design, and the opportunities with regard to Dutch building regulations are described.

Part 2
The first CFD-simulations are made based on a full scale test case presented by Heselden. After a short introduction of the test case (section 2-2), results from the simulations are presented and compared to both the measurement results as well as numerical results obtained by Markatos et al (section 2-3). Some variances on this case are made; results are presented in section 2-4.

Part 3
Section 3-2 introduces the case that is subject to further research; car park Fleerde in Amsterdam. Measurement results as well as formerly obtained simulation results are also presented in this section. Section 3-3 introduces the set-up of the problem while analysis of the simulation results and comparison to measurements and prior simulation is presented in section 3-4. This is followed by results from the alternative cases in section 3-5.

Part 4
Conclusions and remarks are gathered in section 4-2. Recommendations are made in 4-3, primarily based on results obtained but also initiated by experience gained during the process.

Appendices
In the appendices, additional material is included as only main results and most important graphics are included in the report. The appendices contain unprocessed data as well as more extensive and detailed information about the various aspects introduced in the report.
Acknowledgements

Fire safety science is not (yet) part of the curriculum in the Master Building Physics and Systems at the Technical University of Eindhoven. Although there is certain linking with the more traditional research field within building physics (e.g. heat transfer and climate control) a partner had to be found introducing the details of fire dynamics.

Efectis, the former TNO Centre for Fire Safety, was contacted and a cooperation, in which the graduation process more and more developed, was born. Except for the knowledge, the inspiring environment in the near vicinity of the fire laboratory was a welcome addition in the graduation process. I would like to thank all employees and staff at the Rijswijk facility for their input in my project, both with respect to content as well as process. In particular Tony Lemaire, Leander Noordijk and Yvonne Olij-Kenyon, for their critics and discussion.

A special word of thanks also to my supervisors at the Technical University; Marcel Loomans and Jan Hensen, and several others from the professorship of "Ontwerpen van het binnenmilieu".

Also, I would like not to forget to mention the support from colleagues and staff form Adviesbureau Van Dijke, my employer during the graduation process, which was on part-time track with my first steps in the world of working in the (fire) safety branch. Although the combination sometimes was rather demanding, I still do not regret the parallel track; I benefited from the additional experiences.

Of course, I could mention numerous people in my direct environment, for their support and help of all sorts, both practical and emotional. Especially the latter, with the passing-away of my father still fresh in memory, I appreciated very much. Thanks all!
Introduction

The graduation project comprehends several stages, resulting in a dual goal.

First goal is to reproduce results obtained from the full-scale fire test performed by TNO in car park Fleerde using the CFD-code FLUENT. Therefore, the car park has to be modelled, additional sub models and profiles have to be collated and simulation results have to be analyzed. Measurement results and previous simulation results obtained by utilization of the CFD-code VESTA [1] will form the basis for the analysis.

Second goal is to qualify the influence of some non-design specific variables on smoke flow in car park Fleerde. Three objectives can be distinguished:

1. Reproducement of the smoke movement in car park Fleerde and comparison of the results from the CFD-simulation(s) with measurement data obtained by TNO;
2. Qualification of the influence of several non-design specific modelling parameters (e.g. the number of cars present in the car park) on the smoke movement pattern;
3. Qualification of the influence of several design parameters (e.g. structural beams or flat ceiling) on the smoke movement pattern;

Besides these overall goals, a number of objectives have to be enclosed for practical reason. Some knowledge about the backgrounds on CFD and the use of CFD in fire situation is needed for proper modelling. Both theoretical and practical knowledge therefore have to be developed during the first stages of the process.

Research questions concerning the literature study and practical training

Main goal of the literature study is developing basic knowledge about the use of CFD-modelling for heat and smoke movement. Some questions to be asked are:

- What is CFD, how is it used (for fire and smoke movement modelling)?
- What limitations can be distinguished; numerical as well as concerning effort and the amount of time spent?
- Does use for fire situation modelling require specific input, what methods are available and to what extend is high-detailed modelling necessary?
- What is known about the effects of the design assumptions and model parameters?
- How can CFD-results be evaluated; especially concerning numerical accuracy?

These problems form the main subject for the literature study. The actual list of questions consists of a number of more detailed topics, relating various aspects and providing an overall view on the use of CFD in heat and smoke movement modelling.

Practical training with CFD-simulations is part of the preparatory stage. This is filled in by modelling a relatively simple case, used as an introduction-case. However, this case provided the opportunity to do some research related to the second part of the project; the study of variants. Therefore, a more profound study than initially intended is performed. Results do not give a full overview; instead they provide general information concerning the influence of several modelling alternatives.
Part 1
Theoretical overview
Introduction

During the last decades, Fire Safety Engineering (FSE\(^1\)) became of growing interest in the Dutch building approach to prevent fire and limit consequences; both personal as well as material. Following other countries like Great Britain, Sweden and the United States, the field of FSE more and more interacts with regulations and its presence is embraced by architects and gradually accepted by those entrusted with verifying the building permit.

Traditionally the Dutch regulations concerning fire safety have a non-scientific base. The fire safety standard for already existing buildings has grown over time and for new-to-be-build buildings a more stringent approach is chosen. Starting-point is the manageability of a standardized fire in an enclosure and the overall possibilities for occupants to leave the building before a hazardous situation occurs. Unfortunately, direct use is limited considering the enormous variability of buildings, as a result of advanced technology, that is available. Although the Dutch regulation system is in essence performance-based, which leaves space for FSE, the prescriptive design still limits its use.

FSE comprises several aspects including egress simulation and evaluation, fire development, risk assessment, human factors (e.g. irrational behaviour) and, main topic in this study, heat and smoke movement; all based on scientific considerations and research.

The modelling of smoke movement is one of the possibilities to mark (a part of) the scientific safety limits of a non-common building design. Using this approach, design solutions, in which the prescribed regulations do not foresee, are assessed. However, since modelling is a simplified representation of the actual situation, limitations should be very well known. Otherwise an accepted, to-be-thought safe situation can turn out to be pretty hazardous in a real fire situation.

The modelling of smoke movement still is in development. Over the years, lots of research, verification and validation have increased the use of smoke flow prediction models in building design. In the following sections these developments, and their limitations, are introduced. Special attention is given to the use of CFD, which is the main subject in the graduation process.

\(^1\)According to the International Organization for Standardisation (ISO) the definition of FSE runs as follows:

Fire Safety Engineering is the application of engineering principles, rules and judgement based on a scientific appreciation of the fire phenomena, of the effects of fire, and of the reaction and behaviour of people, in order to:

- save life, protect property and preserve the environment and heritage;
- quantify the hazards and risk of fire and its effects;
- Evaluate analytically the optimum protective and preventive measures necessary to limit, within prescribed levels, the consequences of fire.

[ISO TR 13387-1]
Concerning fire safety, a most thorough situation would be a situation in which no fire could develop at all. Creating such a situation, from economical, esthetical and functional point of view, is no realistic option. Therefore consequences have to be confined as much as possible.

In the Netherlands, regulations concerning fire safety basically fall apart in two main items: prevention of casualties (or in the worst case: loss of lives) and prevention of fire growth to non-controllable sizes. In the Dutch building regulations all chapters in which fire safety plays a role can be derived, more or less directly, to these two items. Prevention of other possible consequences, like environmental damage or loss of property, is partly covered by environmental regulations and fire safety demands from insurance companies.

CFD in fire safety engineering is linked to both main items: smoke movement is directly linked to egress concepts, while fire spread, in which smoke movement also plays a role, can be coupled to fire growth scenarios and effects of (automatically) fire suppression systems.

Since smoke movement is of major interest in this project, the implementation of CFD in egress concepts is discussed more profound.

The hot and toxic gases released by a fire are one of the biggest hazards for building occupants in case of fire. The rate at which smoke spreads is high in relation to for example walking speed, while the smoke thickness and properties disable proper visual orientation and walking speed (figure 2). Starting point for building regulations is the distance healthy people, holding their breath, can walk; therefore limiting the distance to be travelled through enclosures. In practice, in most situations this means that within 30 meters an emergency exit has to be reached, enabling people to leave the threatened zone.

In several situations this approach is not feasible, demanding longer egress routes. Within building regulations this is possible, as long as a similar safety level is verified.

In these alternative egress concepts, two terms are commonly used: ASET and RSET, meaning Available respectively Required Safe Egress Time. In short: the ASET describes the time required for the smoke layer to descend to head height or produce too much radiation, while the RSET involves the time required for building occupants to safely vacate the enclosure or building. In order to provide safe egress opportunities the ASET has to exceed the RSET. In determining the ASET, CFD plays an important role while the RSET mostly is covered using evacuation models (figure 3).
However, the use of CFD is not limited to determination of the ASET. The RSET for example, is influenced by the time that passes before fire or smoke is detected. Late detection decreases egress opportunities and therefore has to be prevented as much as possible. CFD modelling can give insight in the detection response in non-standard enclosures, such as rooms in which air movement is strongly mechanically induced or where smoke detection is placed in regions in which flow entering is not optimal (e.g. between structural beams).

Finally, smoke movement, and especially smoke thickness, plays a role in intervention by fire fighters. In order to perform a quick and non-hazardous intervention or rescue, smoke has to be removed from the fire area. Especially in enclosures situated beneath ground level this is of relative importance; smoke moves up while fire fighters have to move down. CFD plays a role in developing optimal ventilation scenarios in fire situations. Most practical use is known in tunnel fire safety and car park ventilation, for which, for the latter, building demands concerning sight and ventilation are prescribed in Dutch building directives.
2.1 Practical implications; the use of CFD and its anchoring in building regulations

As discussed in the previous section Dutch building regulations concerning fire safety is a (mainly) performance-based system. In essence, only a functional requirement is stated, providing designers possibilities for utilization of specific solutions as long as an equal safety level is reached, and the method utilized is accepted by responsible authority.

For relatively small car parks (up to 1000 m², approximately 40 cars) in essence the prescriptive regulations in the Dutch "Bouwbesluit" are sufficient and no equivalence is to be demonstrated. Larger car parks in which natural ventilation does not provide sufficient air change rates (according to NEN 2443), require specific efforts to keep air conditions within limits.

Fire situations in large car parks form a specific situation, which is described in a code of practice [7]. In addition NEN 6098:2007 (still in design) describes smoke control systems in mechanical ventilated car parks, and CFD is mentioned in relation to the requirements made. The code of practice mentioned differentiates car parks in between 1000 and 5000m² and car parks larger than 5000m². For the latter, after 45 minutes a sight length of 30 meters is required, which commonly is demonstrated using CFD. Apart from a number of basic assumptions to be made in the modelling, no requirements concerning the presentation of results and the level of accuracy to be achieved are included.

In the NEN 6098:2007 (design) three objectives are introduced discussing smoke control in fire situations: limitation of the smoke distribution, sight at the fire location and after-care. Proper incorporation of these objectives results in better possibilities for intervention by fire fighters and improved conditions for search and rescue operations. Also a decreased number of building occupants is exposed to high temperatures, toxic gases and limited visual orientation, increasing the possibilities for a safe egress.

Appendix D of the NEN 6098:2007 includes requirements to the CFD-analysis, like goal and scope, modelling assumptions, resources used and results obtained.

For short periods of time, like in egress situations, exposure to high temperature or radiative heat is considered acceptable. The exact values are subject to discussion; several criteria are enumerated here.

Three criteria for non-enclosed places with a short stay function are commonly used: a maximum radiative heat flux from the smoke layer of 1 kW/m², a temperature beneath 45 °C and a minimal sight length of 100 meter [8]. Other studies specify the in-room maximum conditions to be limited at a smoke layer temperature of 200°C, a required sight length of 30 meters and a minimum smoke layer height of 2,5 meters [10]. Since the radiative heat flux from the smoke layer is significantly influenced by the smoke layer temperature, radiative heat fluxes up to 2,8 kW/m² can occur (head-height) in this situation. International standards limit the radiative heat flux at 2,5 kW/m² and include the toxicity of smoke gases, as a maximum CO-concentration is specified. Also, the sight length to light-reflecting objects has a minimum requirement of 10 meters [11] while in studies concerning tunnel safety, in which the same sight length is retained, a maximum radiative heat flux of only 2 kW/m² is allowed [12].

Fire fighters with full protection can endure radiative heat fluxes up to 40 KW/m², but only for very short time-instances (several seconds, like in flash over situations), as can be concluded form the EN 469 [13]. More realistic values are included in table 1, from the handbook of technical textiles [14].

Table 1: overview of tolerated fire exposure conditions

<table>
<thead>
<tr>
<th>Conditions</th>
<th>Exposure time</th>
<th>Temperature (°C)</th>
<th>Heat flux density (kW/m²)</th>
</tr>
</thead>
<tbody>
<tr>
<td>A. Normal</td>
<td>8h</td>
<td>40</td>
<td>1</td>
</tr>
<tr>
<td>B. Hazardous</td>
<td>5min</td>
<td>250</td>
<td>1.75</td>
</tr>
<tr>
<td>C. Emergency</td>
<td>10s</td>
<td>800</td>
<td>40</td>
</tr>
</tbody>
</table>
3. Properties of fluid, fluid mechanics in fire situations

Before a detailed introduction of CFD and its principles can be given, a more generic knowledge about fluid behaviour, properties of fluids and specific behaviour in fire situations has to be present.

Density

Density and temperature for gaseous fluids are mutual related as described in the ideal-gas law: the higher the gas temperature, the lower the density. Density is a so called intensive property, defined as the ratio of an object's mass to its volume. Buoyancy is the force occurring as a result of density differences.

Except for smoke moving upwards in relatively small rooms, in fire situations the density differences between inside and outside air can have major effects on smoke transport through the building, known as the stack effect (figure 4). Depending on the fire location and height of the neutral plane, smoke can be transported to zones far away from the fire location.

Buoyancy also can have a major effect on stratification in high enclosures. Hot smoke, with low density, forms a smoke layer near the ceiling. While rising, temperature differences between smoke and surrounding air decrease. Therefore, in relatively high spaces, like atria, a smoke layer is formed at certain height. More specific, at a height where density differences become negligible and buoyant forces submerge. This effect is intensified in situations in which the building envelope provides large cooling surfaces, like in glass atria. The smoke layer therefore will only accumulate near the ceiling in rooms with limited height.

Turbulence

The ratio between viscous forces (kinematic viscosity, $\nu$) and inertial forces ($v \cdot L$) describes the degree to which turbulence occurs, represented with the Reynolds number. For low Reynolds-numbers, viscous forces dominate in relation to inertia forces, resulting in laminar flow behaviour. Otherwise, high Reynolds-numbered flows are characterized by irregular flow behaviour: with respect to the laws of physics (e.g. conservation of energy) but random in time and space.

In a fire, buoyancy forces induce turbulence behaviour, and vortices occur over a large spectrum of sizes. The interaction between these vortices cause them to break down, forming smaller eddies. This process is continuous, and ends when eddies become so small that viscous forces dominate over inertial forces. Since energy is conserved in an isolated system, the kinetic energy in the vortices is converted in an alternative type of energy: heat. For fire situations, this heat produced by turbulence is negligible; the diffusive effect of turbulence is not.
Heat transfer

Heat transfer can physically be described by means of three mechanisms: heat conduction, heat convection and heat radiation. Although only convection and radiation play a major role in smoke movement modelling, all three are shortly illustrated in the following section.

*Conductive heat transfer*

Conduction involves the transfer of energy through solids, as a result of molecules vibrating against one another. As density decreases, so does conduction; a smaller amount of molecules is available, decreasing the chance of particles colliding. The relation between temperature difference and rate of heat transfer is described by Fourier’s law.

*Convective heat transfer*

Convection describes the transfer of energy by movement of hot particles to cooler zones in the medium. A difference is made between forced and natural convection: forced convection is, in contradiction to natural convection, not induced by buoyant forces. Instead, forces exerted by fans or pumps, for example, cause the convective flow.

*Radiative heat transfer*

Finally, radiation is the transfer of energy by electromagnetic radiation in the infrared spectrum, due to the object’s temperature. Unlike heat transfer in case of conduction or convection, no medium is required for the transfer of heat. Radiative heat transfer for wavelengths in between 400 and 780 nanometres is accompanied by visible light; above these wavelengths one speaks of infrared, beneath these wavelengths the radiation is called ultraviolet.

Combustion

Combustion phenomena describe the fire development itself. A fuel reacts with oxygen in an exothermic reaction, forming combustion products, heat and light. It is a chain reaction process in which heat release initializes pyrolysis of solid inflammable materials, adding new fuel to the combustion and therefore enabling fire spread. As long as heat, fuel and oxygen are available, fire will be maintained, visualized in the fire triangle.

Smoke

In most fires large amounts of smoke are produced. The hot and toxic gases are hazardous to building occupants and make a quick and efficient intervention by fire fighters difficult. Smoke consists of numerous combustion products like carbon monoxide and nitrogen oxides as well as the unburned particles known as soot.
4. Utilization of CFD for the prediction of fluid behaviour in fire situations

Fire is an extremely complex physical and chemical phenomenon, linking research fields of fluid dynamics, combustion, kinetics, radiation and multi-phase flow effects together. CFD modelling makes it possible to use the basic conservation equations in the modelling of fire phenomena. However, some of the underlying fluid dynamics, turbulence modelling and combustion problems have not yet been fully resolved.

In the ignition phase, the primary role of the simulation model is to predict the pyrolysis rate as a function of the incident heat flux to which the solid material is exposed. Pyrolysis describes the release of combustible gases from a solid material as a reaction to extensive heating of the surface. One of the difficulties is the determination of the actual incident heat flux, which is part of the gas phase solution. In the developed, high-Reynolds number turbulent flows, the prediction of both convective as well as radiative heat fluxes has proven to be difficult [3]. Also, material properties (like for instance soot production; the more soot, the less radiation) influence the radiative heat release.

Within fire safety studies using CFD, two main groups can be specified. The first describes simulation of fluid flow only, in which the solid phase is decoupled from the calculations and the fire is modelled as a volumetric heat source. The second comprehends the modelling of both gas and solid phases, including interaction between both: pyrolysis and the flames itself. The latter, the actual modelling of the fire is demanding since both chemical and physical principles have to be incorporated. In most CFD studies concerning fire safety engineering, the first approach is chosen. With increasing computational power, more and more sophisticated models however include the combustion itself.

After the occurrence of a fire, heat and toxic gasses are likely to spread to regions outside the immediate fire region. Apart from the determination of the quantity and kind of toxic components, the movement equations of heat and smoke, which are influenced by the fire itself, have to be resolved. In particular in the regions near the fire, strong correlations occur between fire modelling and the solutions for smoke movement.

One of the major characteristics of a flow developing from a fire is its buoyancy-driven nature. As a result of large temperature differences between the smoke plume released by the fire and the surrounding air, and density differences therefore occur resulting in upward movement of the smoke. As a result, turbulence is introduced.

As with the modelling of fire, the modelling of heat and smoke movement consists of several aspects for which have to be accounted for. The proportion between these aspects has to be clear, for it is inefficient to preserve a high level of accuracy for some of the aspects, while others are pushed aside limiting the accuracy of the overall modelling.

As discussed earlier, again a balance has to be found between time, computational resources and the effort to be made. First of all, the modelling of the fire itself has influence on the results to be obtained. This closes the virtual circle as discussed in the first section: fire spread influences smoke movement, smoke movement in its turn has major influence on radiation and turbulence which both are correlated to fire spread. Especially in the near fire zone, this is of particular interest.
4.1 Modelling of fluid flow

Numerical models can be subdivided in statistical and deterministic models. Statistical models provide information based on empirical relations determined from previous situations and events. Instead, deterministic models try to give insight based on chemical and physical relations, and the equations derived from these relations. Still, several quantities within these equations are empirically determined. Within the deterministic models, a distinction is made between zone models and field models, which both will be discussed in the following section.

4.1.1 Zone models (in fire situations)

Computational fluid dynamics is the answer to limitations of zone modelling. The basic strategy of zone modelling, as it was used for a long time with considerable success, is the discretisation of the field of interest in several zones which separately can be described using semi-empirical equations for mass, momentum, energy and chemical species. As averaging is the only way to keep problems solvable (based on homogeneity), relatively large discrepancies are introduced during the progress. Another major drawback is the necessity of a priori knowledge of the flow field; zones have to be described in advance, based on experiments or theoretical considerations.

Zone modelling can be used in fluid problems with certain characteristics. First of all, the problem has to result in multiple zones or layers, relatively easy to distinguish from each other. Second, conditions in each separate zone have to be relatively homogeneous since no local conditions within a zone can be integrated in the modelling. Third, the interaction between the zones in the boundary layers has to be known.

First smoke movement modelling approaches were based on the so called zone models. By dividing the enclosure in a limited number of zones (mostly two, hot and cold, sometimes three, including an additional plume zone) and presuming homogeneous properties of the fluid in each of the zones, each zone can be described using a limited number of parameters. By introducing a horizontal exchange layer, exchange between both layers is accounted for. Then, using semi-empirical laws and ordinary differential equations, the parameters of interest can be determined from mass and energy conservation.

In the Netherlands a well known zone model is the so called "vultijdenmodel" [8]. In this dynamic model an approximation is made concerning height (thickness) and temperature of the upper layer, related to approximations for fire growth and entrainment. Like in field modelling, energy and mass balances are solved for a number of time steps, but more sophisticated submodels for e.g. turbulence and radiation are not included.

Although for small compartment fires in simple geometries reasonably results can be obtained, a number of disadvantages should be lined up.

A first and major drawback of zone modelling is the averaging procedure which makes it impossible to include or calculate local effects without (dramatically) increasing the number of zones, and with that, the efforts to be made to obtain a solution.

Second, zone models can only be used in a number of fire situations. Therefore, accurate use requires a priori knowledge of the situation to be expected. This knowledge can be
obtained from experimental setup or theoretical considerations, preferably both, but is inevitable when reasonable results should be obtained.

Field models do have their limitations, e.g. behaviour of turbulent flows still is simplified. In comparison to zone modelling however, better performance in complex, high detailed fire situations can be achieved. In this graduation project, no further comments are made on zone modelling. Instead, field modelling is introduced in more detail in the following sections.

4.1.2 Field models

A first definition on CFD is given in the preface. Using this definition as starting point, the topic will be introduced more profound in the following sections. First, some general considerations are made; in the next section application in fire situations is discussed.

The research field of computational fluid dynamics is based on the basic conservation equations for mass, momentum, energy (and species concentration). By solving a set of nonlinear partial differential equations, more accurate results can be obtained, e.g. transient behaviour, movement due to convection and transport due to laminar and turbulent diffusion.

The aim of computational fluid dynamics is to numerically solve the partial differential equations that govern fluid flows. Three general equations are available, namely:

The mass of a fluid is conserved (continuity);

$$\frac{\partial \rho}{\partial t} + \nabla \left( \rho \mathbf{v} \right) = 0$$

{1}

The rate of change of momentum equals the sum of the forces on a fluid particle (Navier-Stokes, Newton’s 2nd law);

$$\frac{\partial}{\partial t} \left( \rho \mathbf{v} \right) + \nabla \left( \rho \mathbf{v} \mathbf{v} \right) = -\nabla p + \nabla \left( \tau \right) + \rho \mathbf{g} + \mathbf{F}$$

{2}

$$\tau = \mu \left[ \left( \nabla \mathbf{v} + \nabla \mathbf{v}^T \right) - \frac{2}{3} \nabla \cdot \mathbf{v} \mathbf{l} \right]$$

{3}

In which:

The rate of change of energy equals the sum of the rate of heat increase and the rate of work done on a fluid particle (first law of thermodynamics);

$$dU = \delta Q - \delta W$$

{4}

$$\frac{\partial}{\partial t} \left( \rho E \right) + \nabla \left( \rho E \mathbf{v} + p \mathbf{v} \right) = -\nabla \cdot \left( \sum_j h_j J_j \right) + S_h$$

{5}

These equations form a continuous model and discretization is needed in order to obtain a set of discrete counterparts suitable for numerical evaluation. The flow domain is split in numerous subdomains, and for each of these subdomains, known as cells, the discretized governing equations are solved. Since all in- and outflow over the boundaries is approximated and numerically solved, the entire flow field can be build up from the
cells, giving a complete picture of fluid flow over the entire domain containing all subdomains; the grid.

FLUENT, the commercial code produced by Fluent Inc., utilized in this study, uses discretization based on the finite volume method on a collocated grid. The idea is to choose a finite dimensional space of candidate solutions (polynomials, up to a certain degree) and a number of points in the domain, and to select that solution which most satisfies the equations at the given points. Using this method, the numerical solutions of ordinary as well as partial differential equations can be obtained.

Modelling of fluid dynamics is a careful balance between assumptions to be made and the physical descriptions of the problem. Even when all physical quantities can be numerically described, the accuracy of results is limited by computational resources, level of detail in the geometrical input and available time or money; all mutual related. The quality of the result therefore is directly related to the setup of the problem.

The equations for the governing equations can be included in the modelling in their specific form as presented above. However, since combustion, turbulence and radiation cannot be fully numerically described, as a result of lack of knowledge or computational power, assumptions and alternative modelling have to be introduced. In these following sections, alternative mathematical descriptions for these phenomena are discussed. Although FLUENT provides a number of methods for modelling these phenomena, each of these methods is best suitable for a limited range of fluid problems. In each subtopic, the methods available for use in fire situations will be discussed. Depending on computational resources, stability and accuracy (in relation to overall accuracy), one of the methods will be chosen for further use in the project.

Several codes are available for calculations concerning fire situations. Some of these, like JASMINE, FDS or SOFIE are specifically designed for use in fire modelling. Others, like PHOENICS, CFX and FLUENT provide general CFD-software in which the specific conditions for fire situations need to be included in the model set-up.
4.2 Field modelling; turbulent flow behaviour

One way to describe the different field modelling approaches within FLUENT (and other CFD source codes) is by distinguishing the way turbulent flow behaviour is treated by the solver. After a short introduction on turbulent flow behaviour, the three main approaches will be discussed.

4.2.1 General modelling approaches for turbulence

In fire situations, turbulence occurs; characterized by chaotic, stochastic property changes of the flow field as a result of vortices. Although major property of turbulence is its randomness, both in time and place, turbulent motion still has to follow the laws of conservation. Therefore, some mathematical description of the flow is possible.

**DNS**

Generally, there are three characteristic methods for taking into account the effects of turbulence on the flow pattern. The first, Direct Numerical Simulation (DNS), accounts for turbulence by solving the conservation equations in such a manner that all rapid fluctuations can be determined. Therefore, no turbulence model is applied. Its major disadvantage is the very fine grid size needed in particularly the whole computational domain; for wherever turbulent behaviour occurs, the size of the control volumes has to be consistent with the size of the smallest eddies present.

**LES**

The second, and similar technique also involves solving the conservation equations, but solely for the larger eddies, therefore this approach is called Large Eddy Simulation (LES). As soon as vortices decrease in size, an additional turbulence model is introduced governing the remaining small scale eddies. The LES technique is applied in FDS (Fire Dynamics Simulation): a CFD-code which emphasizes on smoke and heat transport from fire.

**RANS**

The third approach applies an overall averaging technique: the instantaneous quantities are written as the sum of the mean value and a value representing the random fluctuations. A distinction is made between mass-weighted averaging (Favre-averaging), and conventional time averaging (Reynolds-averaging). Major difference between these two approaches is the way they threat density fluctuations; Reynolds averaging assumes that all terms involving density fluctuations can be ignored and density variations are only caused by temperature variations, while Favre-averaging does not. In general Reynolds-averaging will do at certain distance from the fire source whilst Favre-averaging better describes near-source combustion situations. For heat and smoke movement, the primary interest in this project, the Reynolds-averaging (RANS) provides satisfactory results; Favre-averaging will therefore not be discussed in detail.

Figure 7: Turbulence and smoke
Since for fire simulations using RANS the standard k-ε model is widely used (and best validated, [35]) it will also be used in this project. A short introduction is given on the standard k-ε model, no other viscous modelling approached are discussed here. The interested reader is referred to the FLUENT manual.

In the above section the method of averaging is introduced in relation to turbulence modelling. Averaging introduces additional unknown parameters in the system of equations, the so called Reynolds-stresses (the exact reason will not be discussed in this report, the interested reader is referred to [3]), additional modelling has to be used in order to close the equations (equal the number of equations with the number of unknown quantities). Again two approaches are applicable: find and solve equations for the unknowns so they will not be unknown, or introduce a simplification. The latter is commonly used, resulting in the use of the eddy viscosity model in which the unknown turbulent stresses are assumed proportional to the mean velocity gradient. In doing so, the task of the turbulence model is reduced to the determination of the turbulent viscosity $\mu_t$, which can be predicted using the standard k-ε model:

$$\mu_t = C_m \frac{k^2}{\varepsilon} \quad \{6\}$$

The equations in the standard k-ε model make use of a set of constants (table 2), which have been determined empirically in simple, specific configurations. In spite of this, they are well established and have proven applicability in a wide range of flow problems.

<p>| Table 2: Set of k-ε model constants (Launer and Spalding [3]) |
|---------------------------------|-----------------|-----------------|-----------------|-----------------|-----------------|</p>
<table>
<thead>
<tr>
<th>Standard k-ε model</th>
<th>$C_k$</th>
<th>$C_\varepsilon$</th>
<th>$C_{\varepsilon 1}$</th>
<th>$C_{\varepsilon 2}$</th>
<th>$C_{\varepsilon 2}$</th>
</tr>
</thead>
<tbody>
<tr>
<td>1,0</td>
<td>1,3</td>
<td>1,44</td>
<td>1,92</td>
<td>1,44</td>
<td></td>
</tr>
</tbody>
</table>

4.2.2 Turbulence in fire situations

The k-ε turbulence model is discussed in general in section 4.2.1. For utilization in smoke modelling situations, some additional remarks have to be made. Since flows occurring in fire regions are strongly buoyancy driven, additional turbulence is produced. This is of primary importance in the application of the k-ε model in the modelling of fire problems, as discussed by Moghhtaderi and Novoshilov and Fletcher et al [3]. According to their studies, not incorporating the additional turbulence as a result of buoyancy results in more uniform thermal layers.

For better use in fire simulations, several modifications to the standard k-ε model have been developed. For example, the addition of a constant G [3] or the specification of the Prandtl number as a function of stratification [4].
Using the standard k-ε model, some shortcomings of this methodology of introducing turbulence in fire modeling are known. At first, as a result of introducing the eddy viscosity concept as underlying assumption for the k-ε model, turbulent stresses are assumed proportional to the mean velocity gradient. This implies turbulence to be isotropic, which certainly is not the case in fire situations.

By not taking into account the additional turbulence accompanying buoyancy driven flow, results of simulations show relatively uniform thermal layers, instead of strong stratification.

Besides this, previous research brought forward some other disadvantages. The standard k-ε model tends to underpredict the width of the plume, which influences temperature as well as volume and density of the smoke. [3]

In the same studies the influence of ceiling jets was taken into account. The entrainment appeared to be overpredicted, exerting the influence on the smoke spread.

All these effects have to be taken into account while interpreting the results when using and validating smoke movement models in which the standard k-ε model is used. In this project, the standard k-ε turbulence model is used since this is widely validated in fire situations. Use of a modified k-ε model could at itself be a subject of study.

4.3 Field modelling: use of radiation models

As for turbulence modelling, the modelling approach for radiation varies between numerical simulation and a highly simplified method and therefore, as discussed earlier, a balance has to be found between accuracy of the results and efforts to be made (e.g. computational resources).

4.3.1 General modelling approaches for radiation

As a rule of thumb radiation should be included when the radiant heat flux \( Q_{\text{rad}} \) is large compared to heat transfer due to convection or conduction. As a result of the fourth power in formula \( \{7\} \), this will occur at high temperatures; such as in fire situations.

\[
Q_{\text{rad}} = \sigma(T_{\text{max}}^4 - T_{\text{min}}^4)
\]  
\[
\{7\}
\]

The most simplified approach accounts for radiation by ignoring the numerical simulation and identifying radiation as a certain percentage of the total heat release. In the modelling, radiation is taken into account (actually; is not taken into account) by decreasing the total (measured) heat release with a certain percentage that is thought to represent radiation.

Other approaches are directly derived from the concept of energy conservation, in which radiation represents a source term. In these situations rays are emitted from the heat source, and for every obstacle certain absorptive, reflective and transmissive values are specified. The rays are traced until they have lost all their energy as a result of absorption or disappear out of the computational domain. This however, is a highly detailed method, and therefore not used very often in smoke movement modelling. CFD-software provides intermediate approaches.
4.3.2 Radiation in fire situations

Especially in the near fire region, radiation is of primary importance. It significantly contributes to the pyrolysis phenomenon and therefore, induces flame spread. Especially in small enclosures, radiative heat transfer plays a role in the occurrence of a flash over; a situation in which all combustible products self ignite due to high temperatures (above approximately 550 °C) or high radiation (above approximately 20 kW/m²). In large compartments, like car parks, flash over situations resulting in homogeneous conditions in the whole compartment will not occur and are of minor interest.

Soot plays a role in the radiative heat transfer in the domain since soot provides a heat transfer mechanism; heat is absorbed by the dark soot particles, which is in agreement with the fact that dark or black bodies will absorb relatively large amounts of heat. Several other components of smoke (like hydrochloric acid, carbon dioxide and vaporized water) also play a role in radiative heat transfer. Unfortunately, the radiative properties of these gases are complex; energy release is described by quantum physics; they absorb and emit energy at distinct wave numbers. For this reason, approximation models are introduced for the description of radiative properties of gases and smoke: narrow-band models, wide band models and the weighted-sum-of-grey-gases-model (WSGGM). The latter is widely used in fire modelling.

With regard to smoke movement, radiation still plays an important role in the modelling. Both radiations from flames, surfaces, as well as radiation from the smoke layer (soot) contribute to the heat transfer in the domain. The hot, smoky layer beneath the ceiling radiates to lower levels. Since the layer is non-homogeneous, re-absorption and radiation in lower layers is significant [22]. This radiative heat transfer will increase with increasing smoke concentration, smoke layer thickness and temperature.

Two radiation models are commonly used in modelling the radiative heat flux to surfaces in fire situations: the Discrete Ordinates radiation Model (DOM, used in the project) and the Discrete Transfer Radiation Model (DTRM). Main difference is the way in which the solution scheme is build-up; ray-based (DTRM) or based upon a field solution to the radiative transport equation (DOM). A more profound discussion would imply high-detailed knowledge about radiative heat transfer, which is beyond the scope of this project.

The DOM-method is chosen for its stability and decreased computational time needed. The absorption coefficients derived from the WSGG-model are used with the discrete ordinates radiation model by summing the solutions of the radiative transfer equation for each grey gas.

4.4 Field modelling: inclusion of combustion phenomena

In numerous CFD-simulations combustion is not taken into account, and fire is modelled using a volumetric heat source. Although this limits the accuracy of the results in the region near the fire, results describing the flow pattern at some distance show higher levels of agreement (as is concluded from several previous studies, no precise comments are made). Basic assumption is a non-spreading fire, since the volumetric heat source in general does not account for an increase of the heat releasing volume (although this is definable using a number of volumes of increasing size in the model). Since several types of fires provide quasi-steady burning rates after some initial period, it is indeed possible to model the heat generation rate with a reasonable degree of accuracy; either as a constant or as a known function of time.

In fire safety, one of the most important limitations for detailed combustion modelling is the insufficient knowledge of material behaviour in fire situations. Besides this, modelling of combustion requires knowledge of the gas temperature, species concentrations and
amounts of soot in the mixing zone and near region. The modelling has to encompass the chemical reactions -accelerated by the temperature available- and the turbulent mixing in the fire zone. For completeness: two methods of combustion modelling can be distinguished: the fast chemistry conserved scalar approach and the flame let combustion models; they both know their own simplifications, submodels and limitations.

As heat and smoke movement in relation to safety concepts will be the subject of major interest in this study, combustion itself will not be included in the modelling. Instead, a volumetric heat source is introduced.

4.5 Field modelling: inclusion of smoke properties in relation to sight

As discussed, smoke is the collective term for all combustion products (gases, heat) and soot produced by the fire. Some literature describes smoke as a mixture of these combustion products with entrained air, which is in practical cases a more true description.

Dependent on the level of detail desired, various modelling methods are available in CFD. When the actual combustion is integrated in the modelling process, the chemical species can be determined in relative detail. This is interesting in relation to fire spread as well as in relation to egress possibilities, which are to reasonable degree dependent on the smoke compounds produced (toxicity or irritation).

Although smoke particles have little influence on the flow pattern, except for those components playing a role in radiative heat transfer, smoke is taken into account in most simulations. The concentration of particles is directly linked to sight and ventilation effectiveness; both parameters are of great interest in practical situations.

While convective forces acting on the soot are dominant near the fire region, other forces are relatively small. In this approximation specific characteristics of the soot, like mass, size or weight, are negligible in the modelling. However, in practical situations the size and mass of the particles do play a significant role in the sight obtained; a higher amount of soot particles of small size will provide less sight than a small amount of relatively large soot particles with the same total mass.

The Dutch approach regarding smoke and visibility calculations uses some other quantities than standards in other countries. Therefore, some definitions and relevant concepts, all concerning the Dutch approach, are given here.

The optical density (OD, [1/m]) is a quantity determined from the decreased light intensity of a beam travelling through smoke. Although the law of Beer-Lambert includes the distance travelled, the ratio itself does not include the length scale.

\[ OD = k \cdot C \]  \hspace{1cm} \text{(8)}

In which \( k \) represents the extinction coefficient per smoke concentration, the latter represented by \( C \).

The optical density is determined by a light intensity measurement providing input for both following formulas.

\[ 10^{-OD} = \frac{I}{I_0} \]  \hspace{1cm} \text{(9)}

or, in opposite form:
\[
\frac{I}{I_0} = 10^{-z \cdot k \cdot C} \quad \{10\}
\]

(in which \( I \) represents the light intensity after certain distance through smoke and \( I_0 \) is the original intensity. The OD in formula 9 contains an additional index \( l \) and therefore the distance \( z \) is still present in formula 10).

The sight length\(^2\) (SL, \([\text{m}]\)) relates the visibility of light-emitting and light-reflecting objects to the observer's distance from these objects. Visibility is, apart from the negative effect of smoke, also dependant on some more subjective aspects like contrast, colour, physiological effects and stress. Experiments have resulted in the following rules of thumb:

\begin{align*}
\text{SL} &= 2,5 \cdot \frac{1}{\text{OD}}, \quad \text{for light-emitting objects} \quad \{11\} \\
\text{SL} &= \frac{1}{\text{OD}}, \quad \text{for light-reflecting objects} \quad \{12\}
\end{align*}

which is the distance at which an 99.68% intensity decrease is perceived and only 0.32% of the light is transmitted;

which is the distance at which an 90% intensity decrease is perceived and only 10% of the light is transmitted;

This approximation is based on the assumption that homogeneous conditions are met in the smoke layer. When postprocessing results using specific, local values for sight length, these values have their origin in the assumption that the local conditions are met in the whole domain.

This assumption limits the use of the detailed information available from the simulation so probably a more advance evaluation is desirable. A postprocessing calculation can be included, integrating the consecutive light obstructing smoke values, the extinction coefficients, over a certain differential path-length, \( ds \), resulting in the actual sight length in a certain direction.

\[
\frac{I}{I_0} = \exp \left( - \int_s^z k(s) ds \right) \quad \{13\}
\]

In the latter approach, the sight length from a point in certain direction is based on a more true description, in which the impact of local increased or decreased sight length within the total line of sight is taken into account. One has to keep in mind that, by doing so, a probably more true assumption is made, but simultaneously some of the safety margins are removed. A local reduced sight length no longer results in direct disapproval of the situation, which makes the impact of an inaccurate simulation larger.

In addition, some other quantities are of interest for postprocessing the results from the CFD-simulation. In that context, the smoke potential is introduced.

\(^2\) Sight length is a non-common term in international articles but refers to the Dutch term "zichtlengte"
The production of smoke is in practical situations coupled to the production of heat, since
the developing fire will produce both in larger amounts as it grows from initial stage to
full burning stage. The amount of burning surface therefore can be linked to both heat
output as well as smoke production. For most solid burning materials, a smoke potential
of 200-400 m² per kg burned fuel is common. The unit of m³/kg is diverted from unities
for decreased light intensity and volumetric production, namely m⁻¹ ⋅ m³/kg. The term
potential is used since in test facilities the material under investigation is (almost) fully
burned, but dependant on the burning conditions (from which the ventilation condition is
the most important) a higher or smaller amount of smoke is produced. The smoke
potential is no hard material property, but always has to be linked to the burning
conditions present in the testing.

The international standards use the soot yield, instead of smoke potential. This approach
differs from the Dutch approach in that it is based on the hypothesis that the extinction
coefficient of smoke in an overventilated fire is nearly universal. The basic assumption is
that soot from all flames is primarily carbon, with a primary sphere size much smaller
than the wavelength of light and with a fractal dimension less than two [9].
5. Proper use of CFD, limitations and validation

Modelling of fluid dynamics is a careful balance between assumptions to be made and the physical descriptions of the problem. Even when all physical quantities could be numerically described, computational resources, level of detail in e.g. the geometrical input and available time or money, limit the accuracy of the results; all mutual related.

Important part of the use of computational fluid dynamics is the process of verification and validation. Verification implies the looking for errors in the implementation of the models ("are we solving the equations right?"); validation is the checking of the model itself for is adequateness in the specific practical situation ("are we solving the right equations?").

Verification falls apart in two parts; verification of the code and verification of the calculation. Objective of the first is to find errors in the code, the latter tries to determine the accuracy of a calculation. For proper verification detailed mathematical and programming knowledge is indispensable. It is no part of this project.

Validation determines whether the model (conceptual as well as the model implemented in the CFD-code) and computational simulation agree with experimental data. When possible, it is recommended to use experimental data obtained from tests under laboratory conditions. However, in full-scale fire situations like car park Fleerde laboratory conditions are hard to meet, influencing the possibilities for proper validation.

Table 3: overview of experimental data versus results from simulations

<table>
<thead>
<tr>
<th>Experiments</th>
<th>Simulation</th>
</tr>
</thead>
<tbody>
<tr>
<td>Quantitative description of flow phenomena using measurements:</td>
<td>Quantitative prediction of flow phenomena using CFD-software:</td>
</tr>
<tr>
<td>For one quantity at a time,</td>
<td>For all desired quantities,</td>
</tr>
<tr>
<td>At a limited number of points</td>
<td>With high resolution in space and time;</td>
</tr>
<tr>
<td>and time instants;</td>
<td></td>
</tr>
<tr>
<td>Error sources:</td>
<td>Error sources (3 types):</td>
</tr>
<tr>
<td>Measurement errors,</td>
<td>(Conceptual) Modelling (model ≠ reality),</td>
</tr>
<tr>
<td>Flow disturbance by the probes;</td>
<td>Discretization and iteration (errors in numerical model),</td>
</tr>
<tr>
<td></td>
<td>Usage errors (input errors);</td>
</tr>
</tbody>
</table>

Except for errors, also uncertainties are introduced in the simulation of fluid flow. Since no full mathematical description of all fluid phenomena (especially some of those occurring in fire situations, like turbulence) is available, potential deficiencies are introduced.

In this project several methods were used to give insight in the overall accuracy of the solutions obtained from the simulations. Although no full convergence (the state in which subsequent calculation results show hardly any difference) was reached within each time step (due to computational power and time restrictions), the scaled residuals are presented, providing information about the iterative convergence. Also, balances of fluxes should show conservation within the domain and a short study concerning grid dependency and time step size influence is part of the strategy.
Part 2
Case: 2D Room-fire
1. Introduction

In the first part of the project a study concerning a compartment fire in a shopping mall is made. The geometry of the room, the submodels and the additional controls, are relatively simple which makes it an interesting case for the first simulations in preparation for the car park in the second part of the project. This case is used by Efectis in former CFD-studies and no FLEUNT data was available yet. Also, the case provides the opportunity to perform some parameter studies to the geometry and material properties. A short study of variants for these aspects gives insight in effects to be expected in the second part of the project, in which car park Fleerde is the subject of matter.

First, an introduction is given on the full-scale tests performed in the shopping mall, followed by the geometry and modelling in section 2-2. Results are presented in section 2-3, section 2-4 discusses the study of variants. Overall conclusions are collected in section 2-5.
2. Description of the 2D room-fire case

2.1 Description of the original full-scale test facility

The case describes a 3D mathematical model for the movement of combustion products in a room fire. Goal of this case is a sufficient prediction of velocity and temperature distribution in the enclosure.

In 1971, fire tests were carried out by Heselden, in a shopping mall prior to demolition. The series of experiments were especially conducted to give predominantly 2D flow, and partially for this reason results from this experiment were, and still are, used in several validation studies.

In the room (9x6x3m) a fire source (0.5x6x2m) is placed across the full width of the room, at a distance of 1.25 meters (middle) from the rear wall. The front wall provides the ventilation opening, through which exchange with the outside environment is possible.

In the experimental set-up, the opening has a full width and variable height, creating various situations. The fuel used in the experiments was industrialized methylated spirit, with a heat output of 2.04 MW, calculated from the weight loss. Flame lengths, stirred up by the fire, extended to approximately half the compartment height.

Both temperature and velocity were measured, using thermocouples respectively water-cooled anemometers. The thermocouples were placed along the centreline of the compartment, secured to 3 columns of 9 thermocouples each, with 0.3m vertical spacing in between. The columns were situated at 2.56m, 5.76m and 8.96m from the rear wall. Only at 5.46m from the rear wall a velocity profile was determined, using 8 anemometers.

Figure 8: Floor plan of shopping mall, Heselden
with 0.4m vertical spacing in between. The output was sampled once every 15 seconds in case of the thermocouples, and once every minute in case of the anemometers. Also, a smoke rake was placed in order to visualize the flow and indicate any multi-layering not detected by the anemometers.

The experimental work accomplished by Heselden forms the starting point for this case. While several CFD-codes already have been utilized (e.g. by Markatos [20]) for reproducing the results obtained by Heselden, this is an attempt to reproduce the case using FLUENT (release 6.2.16).

2.2 Geometry and modelling

For this particular case using the CFD-code FLUENT, only the opening height of 2m is taken into account. More detailed information concerning the measurements performed and previous simulations using CFD is to be found in the original case studies [20].

The dimensions of the compartment are presented in figure 9.

\[\text{Figure 9: Simplified representation of the shopping-mall test facility [meters]}\]

The right part of the domain (the outside environment) is integrated in the modelling for accuracy reasons. The size of the extra volume is estimated in the previous study by Markatos. Some refinement, particularly in relation to the grid size near the in- and outlet boundaries, was subject of previous TNO-studies using the CFD-code VESTA.
The fire source is represented by a volumetric heat source with dimensions 0.5x6x2m and a heating capacity of 272,000 W/m$^3$, which covers the 6m$^3$ using 80% of the original heat release (20% radiation is neglected, methylated spirit). The measurement positions are visualized in figure 10.

Like in the case described by Markatos, three types of boundary conditions are present in the modelling; floors and walls are represented using an adiabatic wall description, both upper and right side of the outside environment are represented using pressure-outlets (backflow turbulence intensity 10% and -length scale 1 meter) and all boundaries in length-direction are appointed to be symmetric.

![Figure 10: Measurement positions (Heselden)](image)

![Figure 11: Mesh Roomfire case at z=3](image)

The 3D-mesh consists of 127,250 cells representing a total volume of approximately 270m$^3$. Since the accuracy of results could be influenced by the mesh properties (e.g. size or aspect ratio), a short mesh study is part of this case. The detailed modelling setup is included in appendix 1.
3. Results

At a height of approximately 0.9 m results for the first simulation show a clear distinction between the upper and lower layer in both temperature and velocity. Here, the temperature gradient is high and velocities approach 0 m/s, indicating a boundary layer in between opposite directed flows. Figure 12 shows both the temperature gradient as well as the velocity profiles.

Figure 12: Temperature (lines, legend [°C]) and velocity (vectors); original case

Note: due to scaling of the vectors, some arrows seem to leave the domain through closed boundaries. In reality, the near-wall flow is visualized (and scaled), clarifying the deviant direction.

Results for temperature and velocity distribution are presented in figure [13a-f, next page], both as continuous lines. Experimental results obtained by Heselden and numerical results obtained by Markatos are included, represented by round symbols respectively a dashed line.

3.1 Development of temperature and velocity

During the first minute the flow field alters significantly, resulting in a near fully-developed flow after 60 seconds. All simulations are performed using the unsteady simulation settings with a runtime of at least 12 minutes. The steady state approach could not handle the temperature jump at the start of the simulation (t=0), resulting in erroneous runs. Transient development lasts up to a maximum 6 minutes, so the flow field can be considered stable in time after full-run. The check on convergence is limited applicable since the flow had to be calculated using the unsteady approach.

The development of the hot layer during the first minute gives insight in the spreading of heat over the length of the room and is visualized in figure 14a-f (page 41).
Figure 13: a-f: Comparison of original results with previous simulation results and measurement data.
Figure 14 a-f: Temperature development during the first minute; original case
Both velocity and temperature profiles are in comparative agreement with experimental data and numerical calculations. However, near the fire source mutual varieties increase. Temperatures tend to be approximately 50 Kelvin higher in the measurement results, but only above the neutral plane. A possible explanation could be that the radiation from the fire is overestimated. Since the total heat output of the fuel (industrialized methylated spirit) is known, an underestimation of the total heat output is not to be expected. In this case, 20% of the heat is said to be radiative heat and therefore neglected. The amount of soot produced, and the amount of heat radiated by the fire are correlated, as discussed in section 1-4.3.2. An overestimation of the amount of soot produced will therefore lead to overestimation of the radiative heat transfer. Since the fuel, industrialized methylated spirit, produces relatively few amounts of soot, probably, the amount of radiative heat loss is overestimated and therefore, the amount of convective heat released by the fire underestimated.

4.1 Balance and convergence

Although measurements show comparative results, a study of balances and convergence gives further information about the accuracy of the results obtained.

The case describes a steady situation, but in first instance the unsteady solver had to be used in order to get a solution. Apparently, the large amount of heat introduced in the start-up of the case (no fire growth is incorporated) gives some problems. As can be seen, no full convergence is reached after 300s; all lines (figure 15a) still show a descending development. Since the scaled residuals dropped to low and therefore acceptable values, a relative accurate solution is already obtained. After 300s the steady solver is used to obtain better convergence. Since conditions in the room meet the real situation, no problems are to be expected. However, again scaled residuals for temperatures and turbulence show high peaks. Several short alternative studies showed no improvement so this process of using the steady solver was stopped. Figure 14-b shows the regions in which convergence criteria are hard to meet using the steady solver.
The overall energy balance using the unsteady solver shows negligible difference: an amount of 1 632 000 is brought into the domain by the fire and the fluxes report shows a heat transfer through the outflow opening of 1 631 987. Although, as discussed, no full convergence is reached, the heat transfer balances show a high level of accuracy. The monitoring of several temperature and velocity points, provides some more information; this is comparable to the information in figure 14 (temperature development over time). The number of time steps is increased to 900 for the original case and all following variants. For every case the time step size is set to 1 second, and 5 iterations are performed each time step.
The study of variants falls apart in three sections. The first discusses the influence of the mesh size in a short mesh study. The second discusses some aspects that probably form the basis for the underestimation of the temperature in the upper layer. Finally, a brief introduction is given to some aspects that could alter the temperature and velocity profiles, like alternative flame height and the absence of symmetry planes alongside. Goal of the second section is to try and identify differences between the real situation and the input for the simulation. Goal of the third section is to investigate the influence of geometrical aspects and properties.

Every paragraph consists of a short introduction, the alterations with respect to the original model and presentation of the results. The latter is combined with a short discussion and finished with a more general conclusion.

5.1 Mesh sensitivity
First of all, conclusions must be drawn concerning the quality of the mesh. Dense meshes, including a large number of small cells, increase computational time and power needed. A coarse mesh negatively influences the accuracy of the simulation results, as does a quick descending (or ascending) number of cells over a small region. The mesh used in this study is based on meshes generated in previous studies concerning the Heselden-case. Although sensitivity studies concerning these meshes are described, a short and relatively simple sensitivity study is performed as part of this case.

In the first simulation, a mesh twice as dense as the original mesh is used. The number of cells is doubled for every edge of the model, which form the basis for the volume mesh. Since the grading type is kept unaltered, the middle cell (double sided grading type) or outer cell is equal to the cell in the original mesh. Relatively, the densest regions are to be found near the locations in the model where high gradients are expected. In the second simulation, the density of the mesh is halffned. Equal to the approach followed in the densified mesh, the number of cells is halved for every edge.
Discussion and conclusions

Results from the simulation in which a denser mesh is used differ from the original results. The temperatures show little higher values; more close to the measurement data. This at first sight is a bit confusing since the amount of energy does not alter, and the graphs do not show regions in which a lower temperature is present then in the original case. By checking the total heat transfer rate, both balances, original as well as denser mesh, show good agreement (in ≈ out, within 0.01% difference).

Discussing the coarse mesh, the major difference in comparison to the original results is the form of the plateau in the graph, which tends to be harder to identify.

Comparison of the results with the original results show that the mesh used in the original case is a bit too coarse. Densification results in changed temperatures as well as changed velocities. Since a too coarse mesh results in indistinct boundary layers the use of a coarser mesh (e.g. limiting computational power needed) is not justified in this situation.
Figure 18 a-f: Comparison densified mesh with original mesh
Figure 19 a-f: Comparison coarsened mesh with original mesh
5.2 Increased temperature as a result of increased amounts of energy in the domain.

5.2.1 Influence of radiation
Discussing the results of the modelling of the Heselden-case, radiation came to front as one of the possible explanations for the differences between both measurement and simulation results. In the original simulation, radiation was taken into account as an amount of the total heat release (20%) no longer available in the convective heat transfer. Thus, the total heat release was decreased by a 20%, in which the 20% represents the energy involved in the radiative heat loss. Main conclusion in the original case is that the results show an underestimation of the temperature, but only above the neutral plane, as can be seen in figure 20a.

![Graph](image1)

Figure 20a-b: Comparison of original results with previous simulation results and measurement data

5.2.1a -10% radiative heat loss
In the first run, the radiative heat loss was decreased to 10%, increasing the total heat release in the volumetric heat source to 306,000 W/m³. Since the fuel source used in the full-scale experiments produced a relatively small amount of soot, the decreased radiative heat loss probably is in better agreement to the situation studied by Heselden. Results, presented in figure 21, actually show an improvement of the temperature distribution at a distance of 5,76m from the rear wall.

![Graph](image2)

Figure 21a-b: Comparison of increased convective heat transfer with original results and other data
5.2.1b +10% radiative heat loss
In the second run, the total heat released by the volumetric heat source was further decreased to 70%, implying 30% radiative heat loss. It is not to be expected that doing so will improve the results of the original simulation. Instead, in future work it can be of certain benefit when analyzing results in which under- or overestimation of heat release plays a role.

Results

![Graphs showing temperature and velocity distribution](image)

Figure 22a-b: Comparison of decreased convective heat transfer with original results and other data

Discussion and conclusion
The velocity distribution in both runs only varies slightly with in- respectively decreased convective heat release. Air of higher temperature has lower density and a rise of temperature in the hot layer increases pressure in the room, forcing exchange with the outside environment. The ventilation surface does not vary, resulting in velocity changes in the upper layer. Since volume and temperature of inflowing air is unaltered, the cool layer keeps its original velocity distribution.

Apparently, the height of the neutral plane is not influenced by the temperature of the hot layer, which implies that no new pressure distribution over the opening is induced.

Larger varieties in total heat release probably bring to front these effects more clearly but prudence is needed. As soon as the flow field is disturbed by additional side-effects, e.g. as a result of increased turbulence due to high velocities, no comparison can be made with the present situation.
5.2.2 Influence of the environmental temperature
As with the total heat release by the volumetric heat source, rising of the inflowing air temperature influences the temperature and velocity distribution in the vertical plane. In the original case, an outdoor temperature of 293 K was accounted for. In the two runs concerning the environmental temperature, this temperature is varied and the results are compared with the original case.

In the run, the environmental temperature is increased to 303 K, an increase of 10 K with respect to the original temperature of 293 K. Both initializing conditions as well as boundary conditions for inflow at the pressure outlet (the outdoor environment) are adjusted.

Results

![Graphs showing temperature and velocity distributions](image)

Figure 23a-b: Comparison of increased environmental temperature with original results and other data

Discussion and conclusion
As with the variant concerning the radiative heat transfer energy is added to the closed system. This energy has to be conserved in order to achieve balance. However, some slight alterations in comparison to the variant discussing radiation occur. Increasing temperature in the upper layer has, in the case of increased convective heat transfer (variant 2), effect on the velocity profile in the upper layer, with increased velocity as a result. In the case of increased environmental temperature, this rise of velocity is not noticeable. Since in essence nothing changes in this variant, this is to be expected but strange enough the upper layer shows a too high temperature with respect to the energy added to the system.

A first check on the balances and convergence does not show abnormalities that could explain the difference. Theoretically, two mechanisms could explain the temperature increase. The first discusses the density in the domain, which could be temperature-depandant. Although the velocities in the domain do not change, decreased density decelerates the disposal of heat from the domain. Using the Boussinesq\(^3\) approach in the original as well as the new case could provide some more information. A second theory discussed the radiation, but since radiation is not taken into account in the Room fire cases, this opportunity is excluded.

\(^3\) The Boussinesq approximation states that density differences are sufficiently small to be neglected, except where they appear in terms multiplied by \(g\), the acceleration due to gravity.
5.3 Altered temperature and velocity profiles as a result of change in geometrical characteristics and properties

5.3.1 Constant wall temperature

In the original case, wall-, ceiling- and floor surfaces are implemented using an adiabatic surface condition. No conductive heat transfer is realized and instead, the temperature at the wall is determined solely by the heat of the air in the near flow field. One alternative run, in which the temperature of the walls, ceiling and floor is kept unaltered at 293 K, not influenced by air temperature near the wall, is performed giving the following results.

Results

![Graph showing temperature profile](image)

**Figure 24a-b: Comparison of constant temperature walls with original results and other data**

Discussion and conclusions

Concerning the temperature profile over the room height, two aspects attract attention: the ratio between upper and lower layer height and the temperature in the upper layer.

In table 4, the heat transfer through relevant boundaries is included.

<table>
<thead>
<tr>
<th>Table 4: fluxes through boundaries</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Original case</strong></td>
</tr>
<tr>
<td>Heat transfer rate [W]</td>
</tr>
<tr>
<td>Floor</td>
</tr>
<tr>
<td>Wall and ceiling</td>
</tr>
<tr>
<td>Outside air</td>
</tr>
</tbody>
</table>

The percentages are relative to the original heat transfer rate through the outflow boundary.

Apparently, the total heat transfer rate is decreased with 2.3%, which explains the increased temperature in the upper layer. The decreased total heat transfer can be explained from the decreased velocity (as can be seen in the velocity profile), negatively influencing the rate of change of the room.

---

4 Conduction heat transfer on a constant-temperature wall face is the product of the thermal conductivity with the dot product of the area projection and the temperature gradient. For flow boundaries, the total heat transfer rate is the flow rate of the conserved quantity. [FLUENT manual]
Furthermore, a relatively cold ceiling influences the temperature profile over the room height at certain distance from the surface. As can be seen in the graph representing velocity over height, in the alternative situation no clear neutral plane is recognizable in the profile. Instead, the profile describes a curve with a bending opposite to the earlier simulations, which makes the profile probably more in agreement with the measurements performed by Heselden. The number of measurement points unfortunately is too little to draw conclusions. The velocity in the lower part of the upper layer tends to be relatively low, while the velocity near the ceiling is increased considerably.

5.3.2 Non-symmetrical walls alongside
Although measurements are performed in a 3D environment, the case presented by Heselden originally is developed as 2D study case, in which a 2D flow pattern is induced. In order to obtain a 2D solution, the walls alongside are represented as symmetrical in the 3D geometry in the original case, resulting in straight lines in both velocity and temperature profiles in across direction.

As part of the study of variants, both walls alongside are included in the modelling as a wall, instead of symmetry plane.

Results

Figure 25a-b: Comparison of wall-as-wall with original results and other data; centre-line of the room (z=3)
Discussion and conclusions
At the centre-line of the room, at a distance of 3 meters from both front and back walls, in comparison to the original case no significant alterations are visible. The cross-cut at its turn shows, as expected, differences in the zone near the walls. The walls now alongside have adiabatic properties, resulting in temperature increase near two- and threefold corners.

Figure 26: Walls alongside as wall (alternative case), temperature (legend [°C]) at centre-line of the room

Figure 27: Perspective view of transverse temperatures (legend [°C]);
\( x = 1.2 - 5.76 - 8.96 \)

Figure 28: Perspective view of longitudinal temperatures (legend [°C]);
\( z = 0.5 - 3.0 \)
5.3.3 Alternative flame geometry

The final variant discusses the influence of altered flame dimensions. By halving the flame height, and doubling the width, an alternative geometry with same volume is gained. The altered geometry is included in figure 29.

Figure 29: Geometry with alternative flame dimensions

All other aspects, including the amount of heat released by the volumetric heat source equal those in the original case.

Results

Discussion and conclusions

From these graphs can be seen that a total different distribution of temperature is achieved over the height of the room. In addition, velocity profiles differ significantly. Apparently, the flame dimensions play an important role in the resulting flow field. The neutral plane, although not clearly definable, descends to approximately half the height it reaches in the original case. In order to drag enough fresh air to the fire zone, the speed increases near the floor. Also, an increased speed is perceivable near the ceiling. The amount of heat removed from the room by way of ventilation does not alter significantly; in the new situation a 1,2% decrease is found.

This probably explains the descending of the neutral plane while no significant temperature altering above the original neutral plane is found. However, since balances have to be obtained, the amount of input-heat has to equal the amount of heat removed from the domain. The 1,2% difference probably denotes a non-converged situation. No full-convergence study s performed in this (first) part of the project. Except for
fluctuations in between each time step, scaled residuals show a horizontal development after 900 seconds.

6. Overall conclusions and discussion

6.1 Original case and mesh study
Both velocity and temperature profiles are in comparative agreement with experimental data and numerical calculations. However, near the fire source differences increase. Temperatures tend to be approximately 50 Kelvin higher in the measurement results, but only above the neutral plane.

The mesh study shows that the mesh used in the original case is a little to coarse. Results from the study in which a denser mesh is used, differ from the original results and show a better equality with the measurement results. Results from the coarse mesh better equal the results from the previous simulation by Markatos; only a less strong identification of the neutral plane is shown.

6.2 Increased temperature as a result of increased amounts of energy in the domain.
A possible explanation for the underprediction of the temperatures above the neutral plane could be that the radiation from the fire is overestimated. Since the fuel, industrialized methylated spirit, produces relatively few amounts of soot, probably, the amount of radiative heat loss is overestimated and therefore, the amount of convective heat released by the fire underestimated. In the basic case, a radiative heat loss of 20% was assumed. The first variant introduces a 10% smaller radiative heat loss, increasing the energy released by the volumetric heat source with 10% (convective heat). Results give a better approximation when compared to measurement data. As expected, an increase of the radiative heat loss (decrease of the energy released by the volumetric heat source) shows an equal effect, but in opposite direction.

Considering the influence of the environmental temperature, increase of the temperature is perceived when the environmental temperature is increased with 10 K. This is to be expected since the amount of energy in the whole domain is increased. In contradiction to the results discussing radiative heat transfer, the increase is perceivable both above and below the neutral plane. Which at its turn in logical; below the neutral plane air properties meet the +10 K outside conditions.

Although the in- or decreased radiative heat loss has some, but minor, influence on the velocity profile obtained this effect is, logically, not perceived in relation to higher environmental temperatures. In the case discussing radiative heat transfer, a slight pressure difference between inside and outside air is introduced as a result of an in- or decreased amount of energy release by the volumetric heat source. Since in the case in which the outdoor temperature is increased no additional pressure difference as a result of in-domain heat release is induced, the velocity profile equals the profile in the basic case. In addition no explanation was found for the phenomena that the temperature increase above the neutral plane (in the case of increased environmental temperature) is larger than 10K; namely up to 20 K. Since no additional energy is brought into the system, and the height of the neutral plane shows no, or at most minor differences with the original height (as does the velocity profile) no evidentional reasoning is found. Probably, a situation of numerical inaccuracy or inadequate convergence has occurred, but neither balances nor monitoring points show such effect.

Results of the previous simulations performed by Markatos et al [20] only show underestimation of the temperature in the hot layer in the case in which the opening height is 2 meters. For the 1.5 meter opening, results are in comparative agreement,
heat transfer, influenced by the amount of soot produced, cannot be linked directly. Simulations using a sub model for radiation, instead of neglecting radiation in the total heat release, can more accurate demonstrate the influence of radiation on the temperature in the hot layer.

6.3 Altered temperature and velocity profiles due to change in geometrical characteristics and properties.

The introduction of constant temperature walls gives a different velocity profile in comparison to original results. Most remarkable is the bending of the profile in the upper layer, opposite to all other curves (original, Heselden, Markatos). Also, the neutral plane tends to form a more sharp distinction between upper and lower layer. From the balances, a new distribution of heat transfer is perceived; a -14.7% due to ventilation, partly covered by the +12,1% heat transfer to walls and ceiling, and a +0,3% to the floor (only near the fire source), both relative to the original amount removed by ventilation. The 2,3% of the total heat transfer not brought back, apparently results in an increase of temperature in the upper layer.

When the symmetry planes are replaced by wall conditions, a different temperature distribution transverse to the room arises. Near twofold and threefold corners, a temperature increase is perceivable, while in the middle of the room the temperature tends to lower (slightly).

Finally, the alternative flame geometry shows a totally different situation in the room. The neutral plane is hardly perceivable and velocities both near floor and ceiling increase. Apparently, the flame geometry is of relatively high importance (in this particular situation).
Part 3
Case: Car park Fleerde
1. Introduction

The last part of the project discusses the main case: car park Fleerde in Amsterdam. In this part of the project additional submodels, to be included in more complex fire situation modelling, are introduced and a more thorough examination of the results is made.

First goal is to reproduce results from the full-scale fire test using the CFD-code FLUENT. Although accurate measurement results in such an environment and under these circumstances are hard to obtain, the main effects of the fire should be reproduced, forming the basis for validation of the model.

Using the validated model as a starting point, the additional goal is to qualify the influence of a number of non-design specific variables affecting the smoke flow in car park Fleerde.

In this study, three objectives can be distinguished:
1. Reproduction of the smoke movement in car park Fleerde and comparison of the results from the CFD-simulation(s) with measurement data obtained by TNO;
2. Qualification of the influence of several non-design specific modelling parameters (e.g. the number of cars present in the car park) on the smoke movement pattern;
3. Qualification of the influence of several design parameters (e.g. structural beams or flat ceiling) on the smoke movement pattern;

Prior to demolition, car park Fleerde was used as full-scale test facility by TNO and Novenco. Jet fan ventilation, already known from tunnel safety, was tested to determine its effectiveness in car park fire ventilation. The first section introduced the basic principles of such a ventilation system and its use in fire situations.

After an introduction to car park Fleerde, and the measurements taken during the full scale fire tests in 1998, the model itself will be presented (section 3). Geometry, mesh generation and the submodels needed for a proper working are part of this section, which is followed by a section discussing the development stages of the model. Results are analysed and conclusions drawn in section 4 and 5 while the consequences in a practical situation, further discussion and remarks finish off this third part of the report.
2. Description of the Fleerde-case

2.1 Description of the original full-scale test facility
Car park Fleerde, in Amsterdam, was used in 1998 to perform a number of full scale fire tests. Primary goal was to determine the proper working of a new ventilation system which used jet fans instead of conventional ventilation. Ventilation with jet fans already was known from fire safety in tunnels, but limited knowledge was available concerning the achievements in car parks or other enclosures with both broad and wide dimensions. Full scale fire tests, combined with CFD simulations using VESTA [TNO], could bring some new insights in the performance of the system under investigation.

![Figure 32: Photos taken during the full-scale tests at car park Fleerde](image)

![Figure 33: Floor plan of car park Fleerde](image)
2.2 Thrust ventilation in car park Fleerde

A major subject of the graduation project is the thrust ventilation used in the car park. In this section, a short introduction is given.

The ventilation concept in car park Fleerde is twofold; exchange with the outdoor environment by the in- and outlet openings and control of the flow directions within the garage by the jet fans.

In a fire situation, the fire detection system controls the HVAC-system. The first few minutes after fire detection, no additional ventilation is introduced. Extra ventilation increases smoke volume and spread, deteriorating egress possibilities. In certain situations, ventilation facilities will be shut down completely in order to maintain horizontal stratification of the smoke near the fire location.

After these first minutes, the HVAC-system is switched to full power, generally in a number of stages. The increased ventilation discharges heat and combustion products from the garage, enabling upstream entrance by the fire department and/or preventing damage to the structural elements since smoke temperature is lowered due to additional entrainment of surrounding air.

In large car parks, the HVAC system itself does not provide sufficient power to expel the smoke front; an upstream smoke-free layer therefore is hard to achieve. In these situations, thrust ventilation supports the regular ventilation system, increasing local air speed and maintaining velocity in the desired direction. The proper working of the system, including thrust ventilation, is dependent on a number of properties, like flow rate, placing and fire location. Thrust ventilation, originally utilized in more easy to determine tunnel situations, in car parks has to be designed in relation to specific situations, originating from worst-case fire scenarios.
2.3 Measurements taken during the full-scale fire tests

During the tests five parameters were measured over (at least) a 30 minutes period. In this project, the measurements taken during the test in which a Renault Espace was burned down were used.

Some short considerations on the measurement methods and results are made in the following sections, discussing:
- Measurement of air temperature;
- Measurement of radiative heat fluxes;
- Determination of heat produced by the fire;
- Measurement of optical density;
- Measurement of the air speed.

2.3.1 Air temperature

Air temperature in the car park was measured using thermocouples, placed in a rigid grid over height and width of the garage. Overall results of these measurements provide good insight in the temperature distribution over time and place. Unfortunately, due to several aspects (e.g. placement partly under the protective gypsum board), near-fire measurements are of lower quality. It is expected that the temperatures measured close to the fire region will underestimate the actual temperatures to certain degree [TNO].

Figure 35: Placement of thermocouples in car park Fleerde, top view

2.3.2 Radiation

Heat radiation from the fire region was measured using Medtherm radiation meters, placed on a distance of 0,7 meters from the burning car. This distance represents the average distance between two cars and therefore gives insight in flashover as a result of radiational heating by the flames. For the validation of the simulation case, this data is of limited interest.
2.3.3 Heat produced by the fire

The rate of heat release can be determined in three ways; by measuring the decrease in weight (combined with known heat of combustion), by determining heat flow to walls (based on known heat transfer to the walls and ceiling) and ventilation openings or by measuring the loss of oxygen in the air with respect to the amount of oxygen in fresh air (21%). Since the second method introduces inertia* in the measurements, the weight decrease is used to determine the rate of heat release. From this data, a rough transient profile of heat released by the car could be computed, resulting in the graphs in figure 36. Based on the assumption that the rate of heat release is approximately 30 MJ/kg, peaks up to 7 MW occur in the transient profile used in the simulations (appendix 2). This value of 30 MJ/kg is kept unaltered throughout the project, while in the TNO-research using VESTA various approximations (20, 25 and 30 MJ/kg) are used. Results from the FLUENT simulation are compared with the VESTA-simulations in which the approximation 30 MJ/kg is used.

*Note: the difference between both graphs (measurement results for calculated heat release based on mass decrease respectively heat flow to surroundings) can be explained as a result of inertia. Weight decrease appears as a direct result from burning material, heat flow to walls is delayed (by distance, air temperature increase or heat absorbed by smoke particles). These influences also explain the less high peaks in the heat release rate in the graph based on heat flow to surroundings.
2.3.4 Optical density
The optical density was measured on various locations using a Maurer photometer. Due to the weather during the measurement sequences, some of the sensors were deranged, resulting in only one location in which all light measurements are of satisfactory quality.

2.3.5 Velocity of air
Local air speed was determined using bi-directional probes, made by TNO, placed in a line downstream of the fire location. From this line, measurement results in a direction alongside the garage are obtained. Measurements were also performed upstream the fire region, but here the influence of jet fan ventilation is overproportionately large, resulting in backflow and fluctuations between measurement points.

Figure 37: Placement of probes in car park Fleerde, top view

Goal of the measurement sequence was to provide insight in the opportunities and limitations of jet ventilation in car parks, as well as the development of a design car-fire for use in numerical modelling of fire situations. For this project, the measurement results will be used for validation of the simulation results obtained using FLUENT. For further commend concerning accuracy of the measurement results, the interested reader is referred to the original TNO-report [1].
3. Modelling of car park Fleerde

Modelling using CFD can be roughly divided in 4 basic steps, which will be discussed in the following four sections of the report; based on the modelling issues in car park Fleerde itself. There will be some overlapping between various aspects, since some of them cannot be discussed individually. Actual result from the simulations of the basic case will be discussed in chapter 4. For completeness, first software code names and versions are introduced.

3.1 Names and versions of software codes used

The geometrical models of car park Fleerde were created and meshed using GAMBIT 2.2.30, released 2004. This pre-processing software, developed by FLUENT. Inc., can export a model, including its mesh, in such a way that it can be read by FLUENT as a case-file. Numerous meshing methods are available and, dependant on the geometry defined, applicable to the volume(s).

After modelling and meshing the geometry, a case-file is set-up using Fluent 2.6.16, released 2005. This CFD-code, developed by FLUENT. Inc, provides for boundary conditions definition, simulations as well as post processing of the simulation results. FLUENT, in contradiction to specific fire simulations codes like FDS, JASMINE or SOFIE, provides both general equations as well as a large number of specific settings for various modelling scenarios. The specific settings for use in fire situations therefore have to be manually implemented.

3.2 Geometry and mesh generation

Creating the various models used in this project, two modelling approaches were used. The first based on journal-file input (top-down), the second using the sweep-face command (bottom-up).

For all models with a tetrahedral grid, the geometry was build before a meshing command was executed; using the journal-file as a text user interface (TUI). Although the GUI, input by using the graphical buttons in the interface, should have provided the same result, the use of a journal file had some advantages in this particular situation. Some additional remarks are made in appendix 6.

All models based on the orthogonal grid are created using the sweep-face command, applied to a basic face in which all components of the car park, including obstacles, are integrated. Due to the relative complexity of the garage, a huge number of volumes is necessary for creating the whole model.

Unfortunately, the sweep-face method proved to be a highly time-consuming and instable manner, resulting in several erroneous and corrupted files. Whether this is caused by limited processor power or limitations (or imperfections) in the source code of GAMBIT, did not become clear. Eventually, the problems encountered during mesh creation resulted in none of these models being finished. In order to keep the project on track, the decision to revert to the tetrahedral grid had to be made. The use of this grid type has some consequences for the numerical performance, which will be less accurate than orthogonal grid types. Further comments are made in appendix 6.
Some elements in the garage are omitted, or brought back in the model in alternative ways:

- The pavements in the garage are omitted, since their height is negligible, especially in comparison to the grid sizes used.
- The floor of the car park contains a gradual slope over a length of 36.6 meters in between \( x = 33.77 \) and \( x = 70.37 \) (the location is comparable to the high density grid zone in figure 38). Instead of modelling the slope, which would give problems in combination with the orthogonal grid to be used, gravity is adjusted in this region. This will be discussed more profound in the next section, paragraph 3.3.4.
- The form of the structural beams is adjusted to the orthogonal structure of the garage. In reality, these beams are tapered; however in the model they have a rectangular shape. The exact influence of this simplification is not determined. It is expected that no significant change is to be perceived since the presence and size of an obstacle is more important than the actual geometry of the obstacle.

The fire location in the Fleerde-case is modelled using a volumetric source in which both heat and smoke are released. As discussed in section B-4.3.3, variances on the room-fire case, the geometry of the volume can have major influence on the flow patterns in the enclosure. A proper design of the source volume contributes to more accurate predictions, especially near the fire source. During the measurements in the car park, approximations concerning the volume and geometry of the flames were made. Based on these visual observations, the geometry and volume for the modelled situation were chosen. Also, the volume is said to be fixed in time, which neglects the fire growth over time.

Cars in the car park are concave volumes, representing boundaries of the domain. No additional properties are imposed to the surfaces: the walls are adiabatic and no-slip conditions on the surface.

### 3.2.1 Mesh properties

For creation of the tetrahedral mesh in the main zone in the car park, a size function with start size 0.2, growth rate 1.25 and size limit 0.7 is applied. The other zones, like the fire location and the additional zones for local velocity correction are meshed using a spatial distribution with an interval size value of 0.3.

The total mesh consists of 865.273 cells, which is nearly three times as dense as the mesh used in the VESTA-simulations. However, the VESTA-mesh was constructed as an orthogonal grid, which is known to produce more accurate results. The effect of the increased density therefore is neutralized by the decreased accuracy as a result of the grid type used. In addition, the high number of grid cells is needed for adaptation of the mesh using the polyhedral grid type available in FLUENT 6.3. A smaller number of cells could result in a too coarse polyhedral mesh later in the project. Unfortunately, the study discussing the polyhedral grid type was never carried out.

In spite of the use of a size function, the properties of the tetrahedral mesh are difficult to control. High cell skewness is hard to avoid and negative cell volumes have to be excluded by trial-and-error. Although the mesh result can be improved, for instance by using a combination of grid types or further increase of the density, the final result in the Fleerde model is a compromise between accuracy, number of grid cells, skewness and maximum grid cell size.

Highly skewed elements are accepted as long as the number of elements is very small and they do not appear in regions in which large gradients are to be expected. Most important: in the results they should not lead to physical impossible results. In future, a combination of tetrahedral and quadrilateral cells can be used to avoid high skewness in the mesh.

A short sensitivity study for the mesh is performed; results will be presented in section 5.1. Detailed mesh information, including mesh quality statistical information, is included in appendix 6.
Figure 38: Mesh generated for car park Fleerde - basic case

:top view and longitudinal at z=21,4 (dissecting the fire location)
3.3 Problem set up

As discussed in part 1 of the report, CFD-software provides several numerical setup options and dependent on the subject of matter, a combination of options provides the optimal flow field simulation for the specific situation. In the Fleerdee case, the following model setup is used:

<table>
<thead>
<tr>
<th>Version</th>
<th>FLUENT 6.2.16 (3ddp)</th>
<th>: 3D double precision, chosen for stability while using tetrahedral grids with relative high skewness</th>
</tr>
</thead>
<tbody>
<tr>
<td>Solver</td>
<td>Segregated solver</td>
<td>: In which the governing equations are solved sequentially</td>
</tr>
<tr>
<td></td>
<td>Implicit linearization</td>
<td>: In which both known and unknown variables in neighbouring cells are used</td>
</tr>
<tr>
<td></td>
<td>Unsteady solver</td>
<td>: Since smoke is a time-dependant problem; 1st order implicit unsteady formulation</td>
</tr>
<tr>
<td></td>
<td>Node based gradient</td>
<td>: Known to be more accurate for tetrahedral meshes</td>
</tr>
<tr>
<td>Viscous</td>
<td>k-ε turbulence model</td>
<td>: Commonly used in fire and smoke movement simulations, constants are already introduced</td>
</tr>
<tr>
<td></td>
<td>Standard wall function</td>
<td></td>
</tr>
<tr>
<td></td>
<td>Full buoyancy effects</td>
<td>: Inclusion of buoyancy effects on ε</td>
</tr>
<tr>
<td>Radiation</td>
<td>Discrete Ordinates</td>
<td>: With 7 flow iterations per radiation iteration</td>
</tr>
<tr>
<td></td>
<td>Incompressible-ideal-gas</td>
<td>: Relating temperature and density (ideal gas law pV = nRT)</td>
</tr>
<tr>
<td></td>
<td>Absorption coefficient</td>
<td>: user-defined</td>
</tr>
<tr>
<td></td>
<td>UDS diffusivity</td>
<td>: user-defined</td>
</tr>
<tr>
<td>Gravity</td>
<td>Included</td>
<td>: -9.81 m/s²</td>
</tr>
</tbody>
</table>

A full overview of the numerical setup is provided in the setup-summary, presented in appendix 7.

By default, FLUENT only handles constant values describing the various boundary conditions. By providing User Defined Functions, FLUENT incorporates the ability to use transient and inhomogeneous boundary conditions in simulations. Since several of these UDFs, the originals as well as those modified for this study, provide commercial value to Efectis, these are not included in the report.

3.3.1 Rate of heat/smoke release

Heat release by the fire is an energy source in the domain, and therefore included as a source in the energy equation. The transient profile describing the heat development is represented by a set of linear equations. An additional modelling step is required to include the profile; the use of an UDF.

In the UDF describing the heat release by the fire, the flow time is an input-variable. Based on the current flow time and the equations provided in the UDF, FLUENT calculates the corresponding amount of energy release, returning this value to the solver. The solver, at its turn, includes the value in the energy equation solved. As a result, the air in the flame volume is heated, and air temperature increases at the fire location.

Smoke inclusion is handled mainly in the same manner, but a more explicit approach is needed. Smoke is introduced as a user defined scalar (UDS), released by the source volume. An additional user defined function (UDF) has to be defined describing the total mass of smoke particles released per second by the source. The release of this scalar by the volume representing the fire shows equal time-dependant behaviour as the energy release by the fire; both are proportional correlated. By defining smoke as an UDS, representing a non-default variable with only a magnitude, physical properties of the
smoke can, although limited, be included. Within the scalar itself, an UDF is introduced
describing the turbulent diffusivity of the scalar.
Similar to the heat release, the smoke release by the flame volume is set in the boundary
conditions of the fluid representing the fire.

UDFs for both heat and smoke production were available, but had to be adjusted to the
specific fire scenario in this project. No adjustments were made to the UDF describing the
turbulent diffusivity (the spreading of a scalar quantity due to irregular turbulent velocity
fluctuations) to be included in the scalar modelling.

For providing plots of optical density when post processing CFD simulations, a number of
quantities resulting from the calculations have to be included in the process. First of all,
the local smoke concentration (C) is a quantity of interest. Also, some specific input
variables, related to the amount of fuel burned, and the amount of soot produced, are to
be included.

For the calculation of the optical density (OD) a smoke production of 0.1 kg per MJ
energy released by the fire is assumed, resulting in a maximum smoke production of 0.7
kg/s (as the maximum energy release is 7 MW) to be released in the volume of 13.93
m³.
Also, a smoke potential of 250 m⁻¹m³/kg (related to the approximation of the heat
release: 30 MJ/kg) is used, resulting in conversion factor for the optical density of 82.5
(exact: 83.3, due to early rounding errors, impact of this error is negligible). This is: the
smoke potential multiplied with the ratio of concentration burned fuel to concentration
smoke released.

Within FLUENT, the optical density is determined using a custom field function:

\[ OD = \rho \cdot UDS_{scalar} \cdot 82.5 \]  \hspace{1cm} \{14\}

3.3.2 Air exhaust at the rear wall
Downstream of the fire, both windows in the rear wall are equipped with an exhaust
system. In normal situations, CO/LPG ventilation is provided but in fire situations the
maximum exhaust capacity is utilized for smoke ventilation. A UDF for incorporation of
the transient profile for the exhaust openings is written in this project.

![Figure 39: Flow rate (exhaust opening) and heat release in UDF](image-url)
3.3.3 Modelling of the jet fans

The jet fans in the garage provide additional propulsion of air and smoke in x-direction (the direction alongside the garage). By introducing two UDFs (a UDF concerning the balance between exhaust and supply air, and a UDF improving the velocity in the jet stream at certain distance from the supply opening) the jet fan is modelled without modelling the fan in detail.

For two reasons, a submodel should be used to describe this fan behaviour. Proper modelling requires a highly dense grid in these regions. As a quick descending numbers of grid cells will influence the accuracy of the simulation results in a negative way, this implies also relatively high numbers of grid cells at certain distance from these beams. The total number of grid cells used in the model will increase dramatically, requiring higher computational power and longer calculation time. The dynamics near the jet thrust differ from the overall behavioural characteristics. Describing the stream with certain accuracy requires an anisotropic turbulence model, instead of the isotropic k-ε model. Besides this, the development of the stream is influenced by some other aspects, like the shape of the exhaust edges and the turbulent intensity in the stream, again requiring high detail in order to acquire accurate modelling.

The use of UDFs avoids detailed modelling of the jet fan (e.g. fully developed free jet properties at the supply opening, including additional turbulence parameters; validated using several sequences of measurements), which is beyond the scope of this project.

Since a variable placing of the jet fans was part of the research performed by TNO, several graphs from the TNO-report show a different number of jet fans in the car park, or a different placing of the jet fans with respect to the fire location or the structural beams. In this study only one of the arrangements is used.

In the car park a total of 10 jet fans have been placed; 4 in a row downstream the fire location and 6 upstream of the burning car, as in the full-scale measurement performed on the Renault Espace. Starting at 2 minutes after fire ignition, which corresponds to t=300s in simulation time, air speed is increased to 10,18 m/s at both jet fan supply and exhaust nozzles. With a nozzle surface of 0,168 m², per jet fan a flow rate of 1,71 m³/s is achieved.

At the supply opening lamellas are placed, forcing the air in a downward direction (10° in comparison to perpendicular). In the model flow directions from the supply openings are adapted to this situation.

The UDF for the on/off-switching of the jet fan had to be written in this project.

Both supply and exhaust of the fan are modelled using a velocity-inlet boundary condition. A situation is created in which a certain amount of air is extracted from the domain while at an alternative location extracted air is replenished. Therefore, between exhaust and supply openings of the fan, a coupling had to be made accounting for conservation of energy and mass within the domain. Both temperature and smoke density conditions at the exhaust opening therefore were enforced at the supply opening. These parameters are determined from averaged temperature and smoke densities at the exhaust openings, both proportional to the flow rate.

A similar model was made available by Efectis; in addition the model first had to be adjusted to the higher number of jet fans and second to the jet fan line-up in two directions (x and z), instead of a line-up in only x-direction.

As said before, the properties of the flow from the jet fan are hard to control without using various sub models for turbulence and detailed fan properties. In the model, no such additional high detailed models are integrated. However, a simplified approach is chosen, in which local air speeds at certain distance from the supply nozzles are
measured and used as input for a fixed-velocity zone in the model. This increases the accuracy of the velocities to certain degree, both up and downstream of the fixed-velocity-zone. The overall velocity accuracy in the car park also will be positively affected since there is control over a relatively large part of the jet stream contrary to control only at the supply nozzle. Unfortunately, turbulent properties still will not meet the in-situ situation.

For all jet fans an additional correction is made concerning the velocity at a distance of 2.5 meters from the supply opening. Locally, the x-velocity is prescribed in a vertical face (approximately 1.3 m²) placed just beneath the first structural beam downstream of the jet fan. Results obtained from measurements at the jet in car park Fleerde, provide velocity data for this face. The velocity correction was made using a fixed-value fluid boundary condition, linear developing over the height of the zone, in which the measured values for velocity in x-direction were implemented (figure 40).

In the original VESTA-model, these measurement results were compelled to 27 different virtual planes, to which both mesh size and number were adjusted. In this project the above-presented measurement results are extrapolated and rewritten to a set of functions, describing the velocity field in the virtual plane. By doing so, the mesh size used is made independent from the x-velocity field in the plane and large velocity jumps at the edges of each cell are prevented. It is discussable which of both approaches introduces the largest error.

The UDF for the velocity correction had to be written for this project.

![Figure 40: Fixed velocities over the height of the velocity fields](image)

:at 2.5 meters from the jet fan supply opening, in negative X-direction; the points denote measured values [1998, TNO], the line describes the prescribed profile in FLUENT
Whether the virtual face is large enough to encompass most of the flow, is concluded in the original report from the measurements taken in Car Park Loefzijde in Den Bosch as well as from the measurements and theoretical modeling of the free jet stream.

Figure 41: Visualization of the fixed velocity correction

Figure 42: Visualization of the fixed velocity, enlarged; vectors are scaled

Figure 43: Longitudinal intersection over jet fan and correction

Figure 44: Visualization of the fixed velocity at z=21.4; vectors are scaled
3.3.4 Gravitational correction

As discussed in section X, the floor of the car park contains a gradual slope over a length of 36.6 meters. Although no actual slope is integrated in the Fleerde-model, the effects in relation to buoyancy are accounted for. An alternative approach for the modelling of the slope is introduced, i.e. a correction of the gravitational momentum in these sections of the garage in which the modelling of the slope is omitted. In figure 45 the principle of the correction is introduced. The correction of the gravity is included in the modelling in the form of a UDF describing a momentum source, in both x- and y-direction. This UDF is coupled to all fluid zones in the zone 33,77<x<70,37.

\[ \begin{align*}
g_x &= \sin \beta \cdot g \cdot \text{grav} \\
g_y &= \text{grav}^2 - g_x^2 \\
g_{x,y} &= g_y' \\
g_{x,y} - \text{momentum corrects for } g_x \\
g_y' - \text{momentum corrects for } g_y'
\end{align*} \]

**Figure 45: Principle of the gravitational correction in substitution for modelling of the slope**

In comparison to the real situation, in the model the impact of the gravity (y-direction) is too large, while the horizontal component introduced by the slope is neglected. By manually introducing both the horizontal component and an additional upward force, both depend on the degree of incline, it is possible to represent gravity in a more correct way without modelling the slope in the geometry.

This procedure introduces a correction which is independent from the gravitational vector prescribed in the operational conditions panel in FLUENT. The latter is directly referred to in the conservation equations \{2\}.

A similar model was made available by Efectis; in addition the model is rewritten to a model in which the angle of the slope is one of the input variables, instead of the manually derived angle.
The visualization of the effect (figure 47 and 49) is made without additional inducement of velocity in the car park. This means that UDFs for jet fans or air exhaust were not included in this particular modelling. As both situations are relative to each other, the actual temperatures and velocities are of minor interest and no legend is included.

Figure 46: Slope in the car park, approximately 3.4° incline (scaled figure)

Figure 47: Temperature in fire plume, without gravitational correction

Figure 48: Velocity magnitude in fire plume, without gravitational correction

Figure 49: Temperature in fire plume, with gravitational correction

Figure 50: Velocity magnitude in fire plume, with gravitational correction
3.3.5 Radiation and wall conditions

As discussed in section 1-3.2.2, the model used for the inclusion of radiation is the DOM, completed with the WSGG-model for the absorption coefficients of the smoke. The UDF describing this model is based on coefficients from Fusegi and Smith, and is made available by Efectis. In contrary to some other UDFs already discussed, this UDF did not need any adaptation to the specific situation in car park Fleerde.

Wall properties at the floor, ceiling and walls are included using an UDF based on the 1-dimensional heat transfer model, which is derived from the finite differential method including an Euler implicit time discretization. No adjustments are made to this model in this project, except for two altered numerical input variables: for the concrete walls a heat capacity of 2 MJ/m$^3$ and a thermal conductivity of 2 W/mK are used.
3.4 Initialization and solution

Before the simulations are started, all components (e.g. the UDFs) have been checked in the final configuration for their proper working and stability. Although small-scale testing per UDF is accounted for, supplementary information concerning inertia, overall balance and convergence can be obtained in a set of rough first simulations.

Results from rough simulation provide information about the impact of the UDFs in terms of convergence and imbalance, especially at start up. This information can be used to improve the performance of the model since a better estimation can be made concerning need for temporarily increased number of iterations when new components are introduced. When one or more components (like jet fans or the exhaust at the rear wall) start, or show fast increasing values, there is a risk of inertia introduction when only minor convergence is reached in the following few time steps. When the number of iterations per time step is increased for a short period, the convergence is better and inertia effects are decreased.

Furthermore, some insight is given in the performance of e.g. the coupling of jet fan supply and exhaust openings with regard to continuity and the loss of temperature and smoke. Large differences between the amount of energy or smoke restrained from the domain, and the amount brought back by the supply opening, negatively influence the accuracy of the results.

For these reasons, the development of the final model is split up in several stages, as will be discussed in the next section.

3.4.1 Development stages

The development from mesh to full-working case roughly falls apart in 4 sections. Each of these sections is characterized by the introduction of new equations and UDFs in the model. By splitting up the development, the influence and proper working of the UDFs can be determined in the original configuration. The time step size used in the partial cases is relatively coarse, and the number of iterations per time step small, resulting in modest runtime.

The first rough case computes solutions using only the isothermal components available in the model. Flow velocities, influenced by in- and outflow, jet fans and velocity corrections, are determined over 10 minutes flow time. These 600 seconds are split up, resulting in time steps of 5 seconds. A maximum number of 5 iterations per time step is used. Although introduction of heat, and the accompanying buoyant forces, do influence the local flow velocities, a rough estimation of maximum velocities and velocity spread over the computational domain is possible since highest velocities will occur near the outflow opening in the back wall.

Apart from the above mentioned reasoning, isothermal calculations provide practical information concerning the worst-case scenario in relation to the fire location. From the isothermal results one can determine in which parts of the car park, with the ventilation system performing as in a fire situation, the air velocities are low. It is plausible that smoke from a fire at these locations is hard to control, resulting in a situation in which a downstream smoke free zone no longer is established.

In the second case, the scalar, representing the smoke release by the fire, is introduced. Again, the simulation covers the first 10 minutes, calculating 5-second time steps with a maximum of 5 iterations per time step. From the results a rough approximation can be made of the dispersion of smoke from the fire zone. Again, buoyant forces will influence the exact smoke movement.

In addition, this case brought to front that the UDF linking supply to exhaust of the jet fan needs to be introduced at least one time step after start-up of the jet fans. Otherwise, no flux is available at the exhaust opening, resulting in division by zero in the UDF; an illegal operation. Unfortunately, a slight error is introduced by doing so.
In the third and fourth case, energy (i.e. heat release by the fire) respectively radiation and advanced wall modelling are introduced. With the heat release, buoyant forces occur, making additional gravitational correction necessary to reduce the impact of omitting the slope in the geometry.

Contrary to the first three cases, the fourth case only covers a runtime of 210 seconds. Modelling radiance is a time consuming process and after 210 seconds, 30 seconds after the start of the fire, influence can be perceived to a certain degree and the proper working of the UDFs can be examined. For time-saving reasons, no further rough simulation is accomplished.

From this set-up was concluded that all UDF's were stable and worked as intended. Also, some rough approximations could be made on the requirements for additional iterations and decreased time step size during the first few seconds in which a new UDF or large boundary condition step is introduced.
4. Validation of the simulation results

CFD tends to produce good-looking, colourful figures, even when the solution does not match actual fluid behaviour at all. Results therefore have to be examined and checked carefully, making the results considered credible e.g. for decision making. Validation and verification studies complete the sequence of steps to, dependant on the desired detail, accurate and valid results.

A distinction is made between validation and verification: verification examines for source code errors (programming errors; in comparison to posterior mathematical and physical models) and validation determines whether simulations agree with reality, based on comparison with experimental results. The latter is integrated in the project, the first is skipped since the FLUENT source code is already verificated for use in a number of fire situations, including fire in large scale enclosures like a car park.

The validation performed in this study examines whether the conceptual models and computational simulations agree with situations in real world. Comparison of the simulation results with the measurement results (obtained in 1999 by TNO [1]) makes it possible to identify errors and uncertainties, which are to be quantified as best as possible.

4.1 Convergence and consistency
First of all, the convergence is examined. Convergence can be described as the situation in which sequential solutions to the set of discretized equations show minor changes, indicating that the (still unknown) solution is sufficiently close to the solution obtained. In this project, residual plots are used to obtain information about the convergence reached. One has to keep in mind that the residuals show the amount to which equations are not satisfied, which cannot be directly related to solutions errors. It is known that buoyant flows, like in fires, show large scale time variations, which could be a reason for limited converged solutions. By keeping time steps relatively small, better performance is obtained. Although full convergence is infeasible in the simulation of fire situations, some basic remarks can be made using the results from the residual plots.

Second, consistency is traced; a balance has to be obtained between incoming and outgoing heat and mass, both in the domain itself as in components, like the jet fans (especially the UDF regulating the inlet and exhaust opening).

Extra attention has to be given to possible inertia effects in the results. An additional balance check over sequential time steps is introduced to bring these effects to front.

Overall consistency: conservation of mass and energy
The heat released by the fire should equal the total amount of energy within the domain, namely the increased air temperature and heat removed through walls and ventilation openings. By determining the volume and surface integrals, balances for energy can be examined. Since the modelling as a whole suffers from inertia effects both within the calculations (limited convergence, limited number of iterations within the time step) and within the domain, it is very hard to show accurate balance.
Volume and mass integrals (for total energy, total enthalpy, \( \rho C_p T \) and density) are monitored during all simulations.

Total energy \([\text{J/kg}]\) is the total energy per unit mass, thus the mass-weighted integral for total energy over all available zones results in the amount of energy \([\text{J}]\) (from \([\text{J/kg}][\text{kg/m}^3][\text{m}^3]\)) available in the domain at the end of each time step. This quantity knows a negative value since the domain works as a sink for the heat released by the volumetric heat source.

The over-time graph should result in a development similar to the graph describing the heat release by the fire, especially during the time when heat transfer to walls and by ventilation is relatively small. As soon as heat transfer to walls and by way of ventilation increases, part of the energy in the domain is diverted to these boundary surfaces and the balance should include the fluxes in order to obtain overall balance. Therefore, the difference between the mass-weighted integrals over subsequent time-steps should about equal the netto imbalance reported when evaluating the (surface) heat transfer flux report. When checking this, a difference of about 1% (relative to the total amount of energy within the domain) occurs (appendix 11).

In short: heat source + disposal by ventilation + disposal through walls = energy difference within domain.

This can be visualized in a graph representing both sides of the equation. The difference in between both lines is the error made.

A similar strategy can be followed discussing the mass-weighted total enthalpy and the volume integral for \( \rho C_p T \). Differences between subsequent time steps should about equal the netto heat transfer imbalance (surface, flux), and by doing so, as well equal the mass-weighted difference for total energy. These results were not evaluated to such detail as described upfront, and only used for first appraisal while running the simulations.
Consistency within submodels used in the modelling

Each of the additional components in which some sort of averaging procedure is introduced shows minor deviation with respect to the exact situation. In the jet fans, for example, the amount of heat introduced into the domain by the supply opening is derived from the amount of heat deducted by the back side of the fan. This is an approximation, based on local temperature and flow rate, introducing a minor error.

Discussing the fluxes through supply and exhaust opening of the jet fans, a deviation of approximately 1.4% is perceived for both mass flow and heat transfer rate (appendix 8, relative to the total amount of mass and energy through the opening). A more dense mesh near the exhaust openings, or a mesh with minor skewness, should result in a better averaging, decreasing the relative difference. In relation to other uncertainties, the difference is considered satisfactory in this situation.
4.2 Comparison with previous obtained results (measurement and VESTA-simulations)

In the report a selection of the results is presented. For the full overview the reader is referred to the available appendices.

For the sake of readability, the original presentation structure from the TNO-report is used for presentation of the FLUENT results in relation and comparison to the VESTA-results. Apart from the two methods utilized in the TNO-report, a third structure is introduced in addition in this graduation process. Each of the structures will be presented shortly.

Presentation of the results in contour plots

In the original report, contour and vector plots are used for presentation of the VESTA-results. Temperature, velocity and optical density are visualized for a number of time steps, using five surfaces: \( y = 1.7 \text{m} \) (height) and \( z = 8.25, 19.1, 21.4 \text{ and } 25.05 \) (alongside).

The full set of contour and velocity plots is presented in appendices 11-13, for convenience included in separate documents. Since original numerical data from the VESTA-results is not available for this project, comparison to the VESTA simulation results is limited to visual approximation based on contour- and vector plots (as discussed above). The VESTA-results are not included in this report and the interested reader is referred to [1].
Presentation of the results in graphs; height-based graphs

A more profound comparison between measurement results and FLUENT-data is made based on an additional set of graphs, in which per measurement point both measurement results and simulation results are presented over the height. By doing so, a better approximation can be made of the differences in smoke layer and temperature spread. Also, these graphs provide a better first-sight comparison since results are combined in one figure.

For convenience the filled contour plot is included, providing a visualization of the measurement point alongside the car park, at \( z = 8.25 \) respectively \( z = 25.05 \). The full set of height based graphs is presented in appendix 3.

Presentation of the results in graphs: scatter plots

For quantitative validation, model predictions are compared to experimental measurements with use of scatter plots in which the x-axis represents measured values, while the y-axis represents predicted results. By doing so, a quick overview is given in the relative difference which can be linked to a percentage using the +/- 25-percent reference lines included in the figure. The diagonal represents the situation in which the measured value is equal to the value from the simulation, which implies that for points above the diagonal the simulation overpredicts the quantity at the specific time and place, while any point beneath the diagonal denotes an underprediction.

Since it is impossible to include all results at all time steps for any location, a number of measurement points, at a number of time-steps and a number of heights is used for mutual comparison. In figure 56 the measurement points used are presented, including the colour they represent in the scatter plots.

![Figure 55: Scatter plot](image)

For each of these 6 points, 4 heights are included, namely \( y = 0.85 - 1.45 - 1.85 \) and 2.45. Although these heights cannot be distinguished in the scatter plots, they indeed provide additional information. When all points of same colour are located near each other, a vertical temperature gradient is hardly present in both measurement and simulation results. In situations in which only one set of results (measurement or
simulation) shows a vertical temperature gradient while in the other a homogeneous situations is present, all points are positioned in an almost straight line, horizontal respectively vertical.

A similar strategy is followed in the presentation of the results from the study of variants (chapter 5). Using the same points as mentioned above, a comparison is made between results from the basic case and results from each of the variants. The basic case (II) mentioned is not the case (I) validated using the measurements, but the basic case (II) in which the gravitational correction works in opposite x-direction as in first instance the UDF describing the gravitational correction contained an error. Since all variants were already simulated by the time the error was discovered, the comparison is based on similar gravitational conditions in both the basic case (II) and the variants (see also: page 91). In these plots, the basic case (II) is represented at the x-axis; the y-axis gives the results obtained from the alternative cases. Therefore, every point above the diagonal represents an overprediction; any point beneath the diagonal an underprediction in comparison to the basic case (II).

4.3 Discussion

For presentation and analysis of the results, the threefold structure is used: comparison based on contour plots, on height-based graphs and using scatter plots.

4.3.1 Comparison of the results using the contour plots

Data from the previous simulations performed by TNO-CvB only contains the visualization of the results; no numerical data was available. In spite, a comparison can be made based on these VESTA-plots. Point of attention is the different scaling of legends between TNO-CvB data (exponential) and new results (linear). Similar scaling was attempted but the results from FLUENT were hard to interpret due to the lack of numerical information within the plots. Therefore, a more straightforward legend scaling is used.

All new contour plots are presented in appendices 11 to 13. The contour plots from TNO-CvB are not included in the report [8].

Main difference between the result form TNO-CvB and the simulation using FLUENT is the gravitational correction; included in the new model setup and omitted in the previous modelling. In the results comparison, this also comes to front.

Based on the comparison the following general differences between FLUENT-results and VESTA-results are perceived:

- First, the stratification obtained in the results form VESTA (by TNO-CvB) is less obvious perceived in the new results using FLUENT. The gravitational correction tends to push heat and smoke to the zones upstream of the fire, decreasing uniform layering. In comparison to the results in which no gravitational correction is included, better agreement is achieved, which makes it plausible that some of the differences between VESTA- and FLUENT results have to be referred to this difference. This effect lasts throughout the whole simulation.

- In relation to the first comment, temperatures upstream of the fire tend to be higher in the FLUENT simulation. In the VESTA simulation temperatures within the first two segments (bounded by the structural beams) downstream of the fire do equal or are even higher (t=300). In the FLUENT simulations, in the same time step, the 50°C temperature-isotherm only extends to the first segment downstream of the fire, but higher temperatures (up to 80°C) are found upstream of the fire whereas the VESTA simulation only shows minor temperature increase in the upstream regions. Again, these effects last throughout the full 900s of simulation.

- For the second point: when comparing the VESTA results with results from the variant in which no gravitational correction is included, better agreement is
present. Although temperatures downstream of the fire are overpredicted a little (relative to the measurement results approximately 12%), the upstream fire region shows similar results (e.g. upstream temperatures within 50-60° at t=720s in both VESTA- and FLUENT results).

- An obvious clear difference is the temperature near the ceiling, which is almost everywhere close to 20°C in large parts of the car park when determining the FLUENT results. The VESTA results show a minor temperature drop near the ceiling while the gradient in the FLUENT results is relatively large. No exact results are given since these are highly time and place dependant. Since no (simple) explanation for the difference was found, the wall model is used in unchanged form in all variants. However, in further research the implementation of the wall model should be reviewed.

- When comparing velocity contour plots, both results show relatively good agreement, both in jet stream length (decrease to <1 m/s) as well as velocities near ventilation exhaust and inflow. The airflow direction is hard to determine from the vector plots, which makes it difficult to determine whether the amount of reversed flow in between the jet fans correlate.

- Finally, as with the temperatures, the VESTA results for optical density show better agreement with the results in which no gravitational correction is included; optical density is higher upstream of the fire region when compared to the FLUENT basic case (including gravity).

More detailed descriptions result in large amounts of information concerning minor, time- and place dependant differences, which are less relevant within the context of this study. No further attempts are made to find and describe all deviations.

4.3.2 Comparison of the results using the height based graphs

The previous section focussed on the difference between previous VESTA-results and results using the FLUENT code. In this section a comparison between measurement results and new simulations results, based on temperature, is made.

Figures 55 and 56 show several height based graphs at t= 420 (4 minutes after start of the fire, 2 minutes after start of the jet fan ventilation support). Several graphs show relative good agreement with the measurement data, but at some locations, especially upstream of the fire, the simulation overpredicts the temperature over the full height of the car park. In addition, as already perceived in the previous section, a large temperature drop near the ceiling results in underprediction of the temperature in this zone.

The full set of graphs is presented in appendix 3. In the first few minutes after the start of the fire the majority of points show simulation results near the measurement data and although several locations keep this agreement (mostly points downstream of the fire location) some points show a minor to mediocre overprediction (up to 25% relative to the measurement results) of the temperature in comparison to the measurement data. Especially the region upstream of the fire location suffers from too high temperatures in comparison to measurement data. The jet fans do not provide sufficient support of the ventilation mechanism and spread of heat (and smoke) is not fully limited to the region downstream of the fire.
Comparison with measurements: Height based, basic case with gravitational correction in right x-direction

Figure 57: Comparison between measurement data and simulation results, odd numbered points
Comparison with measurements: Height based, basic case with gravitational correction in right x-direction

Temperature at point 4, t=420 s

Temperature at point 14, t=420 s

Temperature at point 18, t=420 s

Temperature at point 6, t=420 s

Temperature at point 16, t=420 s

Temperature at point 20, t=420 s

Figure 58: Comparison between measurement data and simulation results, even numbered points
### 4.3.3 Comparison of the results using the scatter plots

As already remarked in the previous section, temperature is overpredicted in a number of data points in comparison to the measurement data. In the scatter plots, this overprediction again is noticeable since the majority of data points is located in the section above the diagonal. The relative difference becomes smaller as the simulation proceeds, to be noticed by the larger number of scatter points near, but above, the diagonal at t=840s. This can probably be explained by the decreased overall temperature in the car park, since the peak energy release occurs at 540s and the peak heat transfer through boundaries follows this development with minor inertia. With overall decreased temperature, the relative difference also decreases. However, at t=420s most data points are to be found in the same region (within 20<x<40 and 30<y<70) as is perceived in the scatter plot for t=840s.

![Scatter plots for comparison of measurement data with simulation results](image)

*Figure 59: scatter plot for comparison of measurement data with simulation results*

Unfortunately measurement data for velocity show large fluctuations and the influence of the jet fans is overproportional in these results, especially upstream of the fire, as discussed in section 2.3.5. Therefore, no comparison based on velocity is made.
4.4 Conclusions

Major differences between the FLUENT simulations and both VESTA and measurement data are summarized in this section.

The FLUENT simulation tends to overpredict the temperature in the regions upstream of the fire location; apparently reversed flow occurs spreading smoke to the upstream regions. An effect that, to some extend is perceivable in the measurement data but does not correlate to results from previous simulations using VESTA. In spite, differences become smaller when VESTA results are compared to the case in which no gravitational correction is included.

The temperature near the ceiling is underpredicted by FLUENT, indicating that wall conditions are not in agreement with those used in the VESTA simulation nor those present in the car park.

Velocities within the domain tend to meet the results from the VESTA simulation; no full comparison to measurement data is made since this data suffers from strong fluctuations and therefore is of limited use.

As does temperature, results for optical density from the VESTA simulation correlate best to results from the case in which no gravitational correction is included.

Although the simulation results show a development in the right order of magnitude when compared to measurement data, and main smoke flow principles are met, results to some extend question the accuracy of the model. Especially the insufficient effect of the jet fans in the simulation, unable to keep the smoke downstream of the fire as in the full scale test, does not subscribe a proper validation. Ceiling conditions decrease full smoke layering and a ceiling temperature of 20°C near the fire location is rather extraordinary. The UDF describing the wall properties therefore has to be further checked in, and if needed adjusted to, this particular situation.

However, since the study of variants uses the basic case as a reference, the results from that study are relative to the results obtained from the basic case. This is independent from the conclusion that this basic case fully describes the behaviour during the full scale fire test.
5. Study of alternatives

Main purpose of the project is the examination of the influence of geometry-independent variances on the spread of smoke, and inherent to this, safety for building occupants and/or fire brigade. As discussed in section 2.2, CFD simulations are used to demonstrate an equal fire safety level in comparison to the level aimed for in building regulations. Boundary conditions and simulation methods are, to limited degree, determined in practical guidelines with no juridical basis. Besides this, general agreement has to be obtained with all interested parties, e.g. discussing the worst-case fire location or the influence of wind pressure. Unfortunately, the influence of some modelling aspects is not known in high detail, making assumption-based choices in preliminary consultations inevitable. Goal of this project is to quantify a number of these aspects in relation to smoke spread and safety.

Based on practical experience available at Efectis and results from the Room fire-case, the following aspects are thought to be interesting:

- Apart from the burning car(s), in CFD-simulations concerning fire safety in car parks in general no additional cars are included in the modelling. Probably, the flow patterns in the car park are affected by these obstacles; making modelling of additional cars significant.
- In common CFD-simulations, jet fans are modelled in a relatively easy manner; based on mass flow rate or inlet velocity. Since detailed modelling without the use of sophisticated submodels is impossible, additional effort is sometimes made to enhance the jet stream behaviour at certain distance from the jet fan opening. Probably these additional modelling steps improve the accuracy of the results to such a degree that omitting them is no longer justified.
- In fire situations, structural beams of significant height act like smoke screens, preventing the smoke from dispersion during the first stages of fire development. Later on however, the obstacles introduce additional turbulence, resulting in air entrainment in the smoke layer and therefore increase of smoke volume and decrease of smoke temperature.
- In the Fleerde case an approximation is made of the flame volume; the volume in which heat and smoke are released in the simulation. The approximation describes the full grown fire, but it does not account for the small fire size immediately after ignition. Since the flame volume is thought to influence smoke movement, a good approximation of the volumetric heat source is of relative interest discussing the results.

For the sake of readability each of the variances included in this project is discussed using a report structure equal to the structure applied to the basic case. As a result of limited time, some aspects are not worked through to such a high detail as introduced in the basic case, section 4. Again, scatter plots are presented in which the points above the diagonal represent an overprediction of the quantity of interest in comparison to the results from basic case II. In addition, visual comparison is possible using the appendices in which the iso-plots are presented. Some of these iso-plots will also be included in this report, as visual support to the comments made.

Halfway the project it came to the front that the script-file managing the gravitational correction contained a false x-direction-vector; a slope in opposite direction was created. The basic case (II) had to be reproduced, using the right UDF and resulting in basic case (I) for comparison with measurements (and previous obtained results) would be of limited value using the first set of results (gravity in the wrong direction). All results of the cases concerning the variances (including the same mistake as originally in the basic case) were available by then, and since comparison could be made between the original
basic case and the variances, no attempts were made to reproduce all cases. When interpreting the results, the opposite slope-direction should therefore be taken into account. The original basic case, including the slope in opposite direction, is introduced first. All variances are compared to this case. In short:

The following results are included in the report:

Basic case: All UDFs, correct gravitational correction; basic geometry
No gravity: All UDFs except for those correcting the gravity; basic geometry
Basic case II: All UDFs, gravitational correction in wrong direction; basic geometry
No cars: Basic case II, alternative geometry, without cars
Small flame volume: Basic case II, alternative geometry, smaller source volume
No fixed velocity: Basic case II, no UDF correcting velocity at 2,5m from fan
Coarse mesh: Basic case II, alternative mesh
No beams: Basic case II, alternative geometry, without structural beams

The "basic case" is the case which is compared to the measurement results as discussed in chapter 4.

As for the latter: although a grid study should be included in every CFD case (theoretically, it is part of the basic survey), the alternative mesh is introduced as a variant for the following reason. The case including the alternative mesh is based on the first set of cases, those in which the gravitational correction is in the wrong direction. When it should be presented on its theoretically best place within the context of the research (in section 4), probably lack of clarity is introduced, since the main case in section 4 is simulated based on the improved slope direction. A better understanding and comparison is achieved when the results of the alternative grid are presented in direct relation to the basic case II, in which the gravitational correction still works in the wrong direction. This case is referred to in the following section.
5.1 Alternative (coarse) grid
Grid dependency contributes to erroneous solutions since results do not solely depend on flow conditions, but additionally are influenced by cell size, skewness and growth rate. Grid studies normally provide a number of simulations, with increasing number of grid cells (of decreasing size).

However, in this project an alternative approach had to be used. Using the tetrahedral mesh it was quite a challenge to find an appropriate balance between the number of cells over the height of the car park and the total number of cells in the whole car park. The desirable number of cells over the height resulted in a too high total number of cells in the domain, increasing the overall runtime explosively. In theory: decreasing the grid cell size by a factor 2 increases the computational time by a factor 16 (a factor of 2 for the temporal and each spatial dimension). In order to keep runtime within reasonable proportions, the minimum number of cells over the height of the car park, needed for somewhat accurate results, is used.

In the grid study, no further refinements are examined since no attempts will be made (in case the grid study should reveal the necessity) to incorporate the denser grid in the model; run times will not allow this. Instead a coarser grid is examined; giving insight in the quantities influenced by the grid cell size.

5.1.1 Modifications in the model
A (short) grid study is part of the survey. In comparison to the basic case, the alternative grid is built up from approximately -25% cells (620 513 in the alternative grid, in comparison to 865 273 in the original grid). The size function was modified to a start size 0.29, growth rate 1.25 and size limit 0.7; change was needed for the basic size function (one of the specifications in the method for automatically generating grids) resulted in some negative volumes. The new, coarse grid is presented in figure 60.

5.1.2 Remarks
A coarse grid simulation is known to (sometimes) provide reasonable results for quantities that can be traced directly to the conservation equations of mass and energy (e.g. temperature and pressure); however several other (indirect) parameters will probably show decreased accuracy. For this case, the simulation only covers the first 480 seconds, instead of the full 900 seconds in all other cases. Goal of this case is to determine whether results are affected by the grid, denoting grid dependency. The actual results after 900s, in relation to the parameters of interest like sight length or optical density are of minor interest for this particular variant, and thus will not be discussed in detail. Instead, a more profound examination of a single time-step (t=480s, three minutes after the start of the jet fans) is made, both concerning results and imbalances.
Figure 60: Mesh generated for car park Fleerde - alternative (coarse) mesh; top view at $y=1.7$ and longitudinal at $z=21.4$
5.1.3 Results
The coarse mesh shows several differences compared to the mesh used in the basic case. Temperature tends to be overpredicted using the coarser mesh, as is the x-velocity, as shown in figure 61.

Figure 61: Influence of mesh size at temperatures and x-velocity, data points 10 - 15

Overall balances show equal relative differences compared to the basic case, indicating a comparable consistency. The differences in temperature and velocity therefore cannot be related directly to numerical inaccuracy, but are most likely induced by the altered grid size.

5.1.4 Discussion
In general decreased grid sizes and (thus) increased number of cells positively affect the results in terms of accuracy. However, available time and computational power bound the refinement. In this case both the changed temperatures and temperature gradients (relative differences up to 10% with respect to the maximum temperature and total height) and the increased velocities in the upper part of the domain (relative differences are even higher) imply that a denser mesh (in order of magnitude of the original mesh used in the basic case and all variants) should be preferred in order to minimalize these effects.

Discussing the temperature, a difference up to 10°C is perceived in the upper half of the car park (data points 13 and 15). With respect to temperature gradients, e.g. in data point 12, the 100°C in the original case is found at a height of 0.75m above the floor while this temperature is reached at approximately 0.45m above the floor in the coarse mesh. The velocities even show larger differences, especially data points 12 and 15. Although the height at with a positive velocity (with respect to the direction in the car park) is turned into a negative velocity does not differ in the lower part, in the upper part of the car park both an increased and a decreased height (in comparison to the original mesh) is found.

5.1.5 Conclusion
As discussed, a further refinement is not accounted for since increased time and computational power needed, in comparison to the accuracy obtained in the results, limit the applicability. It is to be expected that a further refinement improves the results. However, the results obtained using the original mesh are expected to provide sufficient accuracy in relation to the goals of the study.
5.2 Alternative flame volume
The source volume used in the basic case is the representation of the flame volume in the full-scale test. However, the volume is based on a visual estimation of the fire size after the growth phase, resulting in overestimation of the flame volume in the modelling during the first few minutes. It is expected that this will lead to decreased temperatures in the near fire region, decreased buoyancy and therefore it probably will influence the smoke movement at certain distance from the volumetric heat source.

5.2.1 Modifications in the model
The model is adjusted to a situation in which a smaller flame volume is available for equal heat release. In comparison to the basic case, the flame volume is reduced to 75% of the original volume. This results in a flame volume of 9,26m$^3$ in the alternative approach. The 75% is not based on specific assumptions but is thought to be sufficient to conclude whether or not the flame volume size affects the results.

As discussed in section 5.1, it is desirable to maintain the grid used in the basic case. By doing so, grid influence is reduced to a minimum, resulting in a clearer base for comparison. The alternative volume is scaled proportional to the original one, resulting in a smaller volume wholly encompassed by the original. This implies a fire of smaller size, with lower flames. Since flame height and surface diameter are proportional related [22], this is an acceptable approximation.

In the UDFs describing heat and smoke release by the volumetric heat source, the rate of heat/smoke release is dependent on the volume, both UDFs are therefore adjusted to the new volume size.

5.2.2 Remarks
Although this approach probably will positively affect the results during the beginning of the fire, a major drawback is the decrease of accuracy in the full-grown fire situation. A more sufficient approach should include a growing source volume, preferably based on modelling of the combustion.

Besides this, several specific conditions in car fire still are not included. One example: As a result of the use of plastic fuel tanks in current car design, fuel will leak from the car after certain time. A fuel fire therefore is to be expected, resulting in increased fire spread and additional heat generation near the floor.

5.2.3 Results
Again, a comparison can be made based on temperature, sight length and x-velocity (figure 65). Results for each of these quantities are presented here for the time steps t=420, t=600 and t=840.

In the graphs for sight length a small number of data points is found. This is caused by the relative large differences (as a result of the logarithmic basis for optical density) with respect to the results from the basic case. Numerical data for sight length is available in appendix 10.
Figure 63: Comparison temperatures basic case and temperatures alternative flame volume

Figure 64: Comparison sight length basic case and sight length alternative flame volume

Figure 65: Comparison velocity basic case and velocity alternative flame volume
5.2.4 Discussion
In the study of variants concerning the Room fire case (section 1-4.3.3), the altering of the flame geometry had relative large impact on the temperature distribution in the room. To certain extend it was expected that a similar effect would occur in the car park. In comparison to the original case the following differences are found:

Both temperature and velocity correspond well to the results from the basic case. Temperatures differ at most 5 °C in each of the specified points (especially in the near-fire downstream region, point 14) and time-steps and all new velocity directions correspond to those obtained in the first simulation.

![Flame volume - sectional view (z=19,1) for optical density at t=420, t=600 and t=840 s](image)

Note: per time-step first the basic case, then the new results; larger views with legend available in appendix 13

In both situations (figure 66) an upstream zone with good sight is found, and although some differences are perceived (especially in the region near the stairwell inflow) in the first few minutes, the situation equals the basic case after starting the jet fans. After a while (approximately 780 s), the zones in which sight length is decreased seem to have increased somewhat in comparison to the original case.

A -25% fire volume does not seem to have large influences on the smoke movement patterns, especially after starting the jet fans. In the near vicinity of the fire minor differences are perceived but the large dimensions of the car park, in relation to the relatively small dimensions of the fire source, seem to decrease the effect.

5.2.5 Conclusion
The volume of the fire hardly influences any of the parameters of interest. Further research is needed to investigate whether this is due to the large scale of the car park.
5.3 Alternative number of cars in the car park
As said in the introduction of part 3 of this report: in numerous simulations the presence of other cars, except for the cars involved in the fire, is neglected. By doing so, a conceivable influence of the presence of these cars on the smoke movements obtained is not taken into account. Probably, the presence of obstacles induces entrainment in the smoke layer or causes local differences in velocity profiles.

The NEN 6098:2007, which is still in design, requires the car park to be modelled without cars, except for the car(s) involved in the fire.

In the basic case, except for the burning car, a number of 39 cars is present in the car park; a degree of filling of 25% is achieved. In the alternative approach except for the burning car, no cars are modelled at all. Since it is expected that the cars increase mixing of surrounding air in the smoke layer (due to increased entrainment as a result of the obstacles resulting in more turbulence in the zone between smoke and fresh air), the overall smoke volume in the car park should decrease when removing the cars from the model. Since increased entrainment (due to obstacles) results in more thin smoke of lower temperature, the resulting layer in the geometry without cars is expected to be of higher temperature. This, at its turn, influences the amount of radiative heat transfer since thicker smoke will be subject to increased radiation within the layer.

5.3.1 Modifications in the model
In the basic case, each car represents a cavity in the domain. A wall boundary condition is applied to the faces bounding these holes. In the alternative approach, each of the cars is filled with a fluid, representing air, and boundaries are switched to interior faces. By doing so, the original grid is maintained, decreasing grid influence. Also, the former location of the cars is more easily to determine when analyzing the results.

5.3.2 Remarks
The fill-up of the car volumes with air implies an increased number of grid cells, especially when the grid refinement at the car boundaries is retained. In the basic case, the obstacles cause altering of the flow direction, which makes grid refinement necessary. In the alternative case, no additional changes in the flow are to be expected; the refinement therefore is redundant and leads to higher runtimes than strictly necessary for accurate results. However, as discussed, for sake of mutual comparison the grid size is left unaltered.

5.3.3 Results
Again, a comparison can be made based on temperature, sight length and velocity. Results for each of these quantities are presented here for the time steps t=420, t=600 and t=840.
Figure 67: Comparison temperatures basic case and temperatures when no cars are present

Figure 68: Comparison sight length basic case and sight length when no cars are present

Figure 69: Comparison velocity basic case and velocity when no cars are present
5.3.4 Discussion

Although in the early stages of the fire the temperature is lower in the simulation in which no cars are included, temperature tends to be equal or even higher in a later stadium, except for results in point 15 (black, in between structural beams enclosing the fire zone). Most analysis points stay within the 10%-margin; again point 15 shows higher differences. Apparently the hot air enclosed by the structural beams nearest to the fire location is diverted more easily as a result of the absence of the cars.

The plots in appendix 11 reveal some other differences. The decreased downstream spread of temperature during the first few minutes after start of the fire (new results) seems to positively affect the egress possibilities during these first minutes. However, this is in sharp contradiction to the increased spread of temperature downstream after the start of the jet fans. The temperature in the upstream region is higher in the results from the basic case, indicating that the absence of cars positively affects the working of the jet fans.

![Figure 70: No cars - sectional view (z=19,1) for temperature at t=420, t=600 and t=840 s](image)

Note: per time-step first (above) the basic case, then the new results; larger views with legend available in appendix 13.

The sight length in the new results shows only a few points in the graph, indicating that differences between basic case and new results are large, or both values do not reside in the scales chosen on the axes. The numerical data, available in appendix 10, shows that only the new results do not fit within the scale included on the axes, the sight length has dramatically improved.

Referring to the plots in appendix 12 the optical density in the downstream region of the fire tends to be somewhat lower in the new results, indicating a larger sight length (which is inversely proportional). Also, the strong gradients found in the results for optical density from the basic case disappear when no additional cars are present in the car park. Higher temperatures downstream of the fire (figure 70), in comparison to the basic case, indicate the presence of hot air (and therefore presumably smoke). However, the sight length is increased. This is a remarkable result since it was expected that the absence of cars would result in fewer amounts of entrained air (due to diminished mixing in the boundary layer), and therefore higher temperatures and decreased sight length downstream of the fire. Concerning temperatures, this hypothesis is followed; sight lengths however show other results.

5.3.5 Conclusions

The absence of cars does affect the results. Since these are related to both time and space (e.g. point 14, downstream of the fire; which shows increased (t= 420s) as well as decreased (t=840s) results for sight length), further research is needed before general conclusions can be drawn.
5.4 Alternative modelling of the jet fans
Jet fan modelling is the next design-independent issue to be researched in this project. In the basic case, a relatively specific approach is used, namely modelling of the jet fan using an additional correction (prescribed velocity) at certain distance from the supply opening. By doing so, a more realistic result is thought to be found, especially at further distance from jet fans and volumetric heat source.

For several reasons, in most CFD-simulations a more simple approach is used. First of all, hardly any situation-specific measurement results, forming the base for the correction, are available. Car-park design (height of structural beams, centre-to-centre distance of the beams) in most cases is correlated to situation-specific structural considerations; limiting the availability of measurement results.
Second, methods for incorporation in the model have to be available in the software.

5.4.1 Modifications in the model
In comparison to the original case, the control of fixed velocities is omitted. The zones in the model are maintained for mutual comparative reasons.

5.4.2 Remarks
Although velocity correction is a more accurate modelling approach, some remarks have to be made. Turbulent properties of the flow through the fixed-value zone are not controlled and flow direction control is limited to the x-direction (y- and z-directions both 0 m/s, no measurement results are available). Especially the latter introduces an error since the jet fan supply opening directs the flow at an angle of 10 degrees, as can be seen in figure 71.

Figure 71: x-direction at fixed-value zone
5.4.3 Results

Again, a comparison can be made based on temperature, sight length and velocity. Results for each of these quantities are presented here for the time steps $t=420$, $t=600$ and $t=840$.

![Figure 72: Comparison temperatures basic case and temperatures with alternative jet fan modelling](image)

![Figure 73: Comparison sight length basic case and sight length with alternative jet fan modelling](image)

![Figure 74: Comparison velocity basic case and velocity with alternative jet fan modelling](image)
5.4.4 Discussion
Temperatures downstream of the fire show similar development during the heating phase, but cooling down of the air tends to be slower in the new results. In the graph presenting $t=600$ s both green and red points lie on the diagonal, indicating no differences with respect to the basic case. Several minutes later, at $t=840$ s those points have moved to the upper left corner of the field, indicating a temperature drop in the basic case while in the new results the temperature sticks at approximately 60 to 70°C. The sight length is hard to interpret in this variant. The absence of the velocity correction tends to improve the sight length, but only in certain regions in the car park. When interpreting the direction of the air movement (with help of additional vector plots, x-velocity with equal scaling and range) some more insight is given.

![Figure 75: comparison of x-velocity vectors, t=900 s at y=1.7m](image)

Note: first (above) the basic case, then new results

All red regions indicate local flow back (positive x-direction). Figure 75 shows relatively large regions of backflow in the basic case in the zones near the wall between both stairwells. Flow back, especially near the end of the simulation when the whole zone downstream the fire is filled with hot smoke, will result in smoke moved to more upstream regions in the car park. In the new results, this backflow is significantly less, which is the major effect of the absence of the velocity correction.

5.4.5 Conclusion
Velocities show, in comparison to the other variants, high differences but that was to be expected: this variant directly influences the working of the jet fans. Since air movement is one of the driving forces for temperature and smoke distribution, it is to be expected that both temperature profiles and sight length are affected by the changed velocity distribution.
5.5 No gravitational correction
Additional gravitational forces are introduced manually, replacing the slope, which is omitted in the modelling as discussed in section 3.3.4, in the car park. In contrary to previous simulations, this simulation does not know this additional correction. The new situation is comparable to a car park in which no slope is present at all.

5.5.1 Modifications in the model
In comparison to the original case, the fluid properties are not corrected using additional up- and sideward forces accounting for the gravity in replacement for the slope.

5.5.2 Remarks
When the Fleerde-case was modelled in 1999 by TNO-CvB, no additional gravitational corrections were made while the slope, as it is in this FLUENT-model, is omitted. Results from this variant study therefore are compared to results from the previous modelling as well as to results from the basic case.

5.5.3 Comparison to results of previous simulations by TNO-CvB
Comparison to results from simulations by TNO-CvB is based on visual approximations. Differences are somewhat hard to interpret since two different scaling types are used, linear using FLUENT and exponential in the results form TNO-CvB. Both plots show accumulation of heat in the near fire region, but temperatures at other points in the domain show some differences. The FLUENT-simulation predicts higher temperatures downstream of the fire, but temperatures upstream (although hard to interpret due to minor legend information due to the scaling-step) show more equality.

Figure 76: Visual comparison of temperature [°C], t=600 at y=2.09m
Upper: FLUENT and lower: VESTA, by TNO-CvB

In appendix 4 the full set of graphs (comparison results with measurement data) is presented.
5.5.4 Comparison to results from the basic case

Figure 77: Comparison temperatures basic case and temperatures with no gravitational correction

Figure 78: Comparison sight length basic case and sight length with no gravitational correction

Figure 79: Comparison velocity basic case and velocity with no gravitational correction
5.5.5 Discussion
Depending on the data point regarded, both under- and overprediction of the temperature occurs; some data point even do no longer fall within the +/− 25 percent reference lines (figures 77-79). The sight length downstream of the fire (point 14, green) as well as in between the structural beams encompassing the fire zone (point 15, black) is deteriorated in comparison to the basic simulation (like the other variants, the case in which the gravitational correction is introduced in the wrong x-direction).

![Figure 80: No gravitational correction - sectional view (z=19,1) for temperature at t=420, t=600 and t=840 s](image)

In the new situation, hot air tends to be stowed in between the structural beams, which act like smoke screens. Both upstream and downstream of the fire smoke leaks to the adjacent segments, while in the original case all smoke was transported downstream of the fire (figure 80). Since the ventilation conditions no longer are supported by the natural smoke flow, smoke is diverted less efficient resulting in decreased sight length and higher temperatures in the near fire region. In the back of the car park (near the exhaust ventilation) the situation improves in comparison to the original case, and larger sight lengths as well as lower temperatures are obtained.

5.5.6 Conclusions
Neglecting of the slope in the car park, both in the geometry or in a submodel accounting for additional gravitational forces, has large influence on the results. Both temperature and sight length are greatly affected by the gravitational forces. Although further research is needed, the conclusion can be drawn that a slope should be incorporated in the modeling.
5.6 No structural beams in the car park
Although the appearance of structural beams in a car park basically is a design issue, the effect of structural beams is part of this study. TNO-CvB performed measurements in situations in which a jet fan was placed beneath a flat ceiling, with no structural beams present. Measurement data regarding the velocity profile at certain distance from the jet fan supply opening was available and could be introduced in this study with relative ease.

It is thought that the absence of structural beams will result in more efficient working of the jet fans: the ceiling emphasizes the flow accomplished by the jet fans instead of disrupting the smooth flow when beams are present.

5.6.1 Modifications in the model
For this simulation several adaptations had to be made in both the geometrical model as well as the submodel controlling the velocity at certain distance from the jet fan supply opening.

In the geometrical model the structural beams had to be removed, jet fans had to be placed directly beneath the ceiling and the zones in which the fixed velocities are enforced had to be adjusted to new dimensions and placement.

The submodel controlling the fixed velocities had to be adjusted to the measurement results obtained by TNO-CvB in car park Loefzijde [8].

5.6.2 Remarks
A new grid had to be defined for the new situation which makes the results vulnerable to grid dependency. The total number of grid cells is somewhat smaller than in the original case (818,859, to 865,273 in the original case), while the total volume of the domain is increased with approximately 75 m$^3$ (in replacement for the structural beams). However, the decreased number of grid cells is expected to be justified since the densification near the (remote) obstacles is no longer necessary.
5.6.3 Results

Again, a comparison can be made based on temperature, sight length and velocity. Results for each of these quantities are presented here for the time steps t=420, t=600 and t=840.

Figure 81: Comparison temperatures basic case and temperatures when no beams are included

Figure 82: Comparison sight length basic case and sight length when no beams are included

Figure 83: Comparison velocity basic case and velocity when no beams are included
5.6.4 Discussion

As expected, heat and smoke is removed from the near-fire zone more efficient, resulting in decreased spread in the cross directions. When interpreting the fluxes through the boundaries, in comparison to the basic case (figure 85) an increased energy output through the ventilation exhausts at the rear end of the car park is perceivable, presented in figure 84.

Figure 84: heat transfer through boundaries with no structural beams

Figure 85: heat transfer through boundaries Basic II
In the basic case heat transfer through the ceiling was the main loss mechanism in the first 600 seconds of the simulation. In the new situation the most important heat loss is provided by the ventilation. Up to 600 seconds, temperature tends to be lower in most of the domain, and sight length is improved. Above 600 seconds, some data points denote improved situation with respect to temperature and optical density, while several other points show worse conditions in comparison to the basic case. As expected, velocities show relatively large differences when compared to the original case. Several points are to be found in a different quarter, indicating changed direction (increased backflow (figure 86, red arrows), especially near the longitudinal wall farthest from the fire).

![Figure 86: Increased backflow (in red) when no structural beams are present](image)

![Figure 87: No structural beams - sectional view (z=19,1) for temperature at t=420, t=600 and t=840 s](image)

Note: per time-step first (above) the basic case, then the new results; larger views with legend available in appendix 13

5.6.5 Conclusion
In general, the absence of the obstacles results in increased flow, explaining the more efficient ventilation in the first part of the simulation. Later on the increased backflow (in opposite direction to the main flow) results in increased mixing and results for temperature, sight length and velocity tend to become strongly time and place dependant. At this stage an overall conclusions is hard to draw. The appearance of backflow in the domain therefore thwarts a clear analysis of the results.
The conclusions from this part of the graduation process will be directly linked to the research question introduced in the beginning of the report:

1. Reproducement of the smoke movement in car park Fleerde and comparison of the results from the CFD-simulation(s) with measurement data obtained by TNO;
2. Qualification of the influence of several non-design specific modelling parameters (e.g. the number of cars present in the car park) on the smoke movement pattern;
3. Qualification of the influence of several design parameters (e.g. structural beams or flat ceiling) on the smoke movement pattern;

Discussion
1. The first item, reproducing the smoke movement in car park Fleerde, pointed out to be quite challenging. Measurement data (obtained by TNO, not in a laboratory environment but in a full scale fire setting), and simulation results (based on a model full of assumptions and simplifications) are compared and show relative large differences in some of the point, especially in the region upstream of the fire. In addition, the simulation results show extraordinary behavior near the ceiling. This indicates that the UDF describing the ceiling properties probably does not correspond to the actual situation in the car park. Despite the differences, the data points downstream of the fire show relatively good agreement, but also tend to overpredict the temperature (a little). In contradiction to the actual situation during the full scale fire test, in the simulation the jet fans were not able to control the smoke movement and create an upstream smoke-free zone.

Temperature, velocity and optical (smoke) density are also compared to simulation results from TNO-CvB, using VESTA [8]. Main difference between both simulations is the additional correction of gravity in the FLUENT simulation, while in the VESTA simulation no efforts are made to substitute the slope, which is omitted in the geometrical modeling. Comparison with the basic case shows differences in the zone upstream of the fire, similar to the differences between simulation results and measurement data. In general, when compared to the measurement results the results obtained by TNO using VESTA show better agreement in the upstream zone and comparable results in the downstream zone. However, when the VESTA-results are compared with the FLUENT case in which no gravitational correction is integrated (like in the VESTA simulation) better agreement is perceived. In the latter comparison, stratification is found similar to the stratification in the VESTA-results. Also, the jet fans seem to achieve an almost smoke free zone upstream of the fire and optical densities also meet previous results.

Mesh sensitivity is examined and results show a mesh sensitivity when the number of grid cells is decreased to 75% of the original mesh. Densification is not under discussion since this, although it may improve the accuracy of the results, is not realistic in terms of time and computational resources.
2. In the second item several non-design specific elements in the modeling have been examined.

First, the volume in which heat and smoke are released is changed, resulting in a volume representing 75% of the original size. This change does not seem to have large influence on the results and only in the near vicinity of the fire minor differences are found. After starting the jet fans these differences also tend to submerge.

Second, the number of additional cars in the car park is decreased; no cars, except for the car on fire, are present. Results show some differences in comparison to the original case. Hot air enclosed by the structural beams nearest to the fire location is diverted more easily as a result of the absence of the cars and the jet fans seem to work more efficient in the new situation. Sight lengths show relatively large differences, in which the conditions downstream of the fire are improved and strong gradients no longer occur.

Third, jet fans are modeled without correction of the flow at certain distance from the jet fan supply opening. As expected, velocities show relatively large differences in comparison to data obtained from the basic case. Back flow of air, especially when the simulation advances, is significantly less in the new results. This probably explains why several points downstream of the fire denote slower temperature drop in comparison to the results from the basic case: less air is moved from the zone downstream of the fire to the zone upstream of the fire and overall velocities have decreased, both limiting the spread.

The effect of the gravitational correction is already discussed in relation to the basic case in the first item.

3. The third item only discusses one (design specific) modification in the model: the structural beams are omitted. As a result smoke and hot air are removed from the near fire zone (and the car park in total) more efficient. Higher heat transfer through the exhaust opening is found (In the basic case heat transfer to the ceiling was the main loss mechanism) and cross directional velocities have decreased.

Two remarks have to be made discussing the results of the alternative study:

First of all there is a certain chance that, instead of determining the sensitivity of the model for a certain aspect or modelling method, the sensitivity of the software itself is resolved. Especially those aspects influencing the numerical solution algorithms are subject to this problem. Performing additional measurements in variant situations can help validate the model, and modelling the variances using alternative software probably gives insight in software-related phenomena.

Second, some results may be geometry-dependant to a certain degree, limiting the overall applicability of results. The introduction of additional cars in the model, for example, probably will have more influence in flat-shaped car parks. Additional geometries should be examined, preferably with measurements results available, before overall conclusions concerning the influence of those aspects subject of study can be drawn.

Overall discussion with respect to the use of CFD

Other than the specific conclusions for both cases, some more overall conclusions concerning the use of CFD, and more specific in fire safety engineering, can be drawn. As can be concluded from the studies of variants, relatively minor alterations in the model do affect the results of the simulation.

As the set-up of both geometry and numerical conditions do include several simplifications and estimations, the simulation is sensitive to input specification. In essence this can be concluded for all modeling: "rubble in is rubble out". Since CFD tends to produce nice looking colorful pictures representing a completely wrong solution (as happened several times during the project), simulations should performed as well as results should be judged by someone able to fathom the basic principles of fluid flow and
thermodynamics and the possibilities and impossibilities of CFD-software. During the project I got a first insight in these aspects but for me, there still is an awful lot to learn.

The sensitivity of the model to input specifications also makes profound validation of the model inevitable. Specific problem when using CFD in relation to fire safety engineering is the availability of proper measurement results. Not only do high temperatures and the strongly turbulent flow near the fire region thwart the measurement itself, also the measurement equipment is vulnerable to these extreme conditions: the deviation increases as calibration of the equipment generally takes place in standard room conditions and in the near fire region additional protection of the equipment for both heat and corrosive gases is needed. In addition, full scale fire tests do damage the structure of building exposed to the fire conditions. This makes it nearly impossible to perform control measurements in situations in which CFD is used to estimate the effectivity of fire safety solutions. And in case a fire occurs, only the remainings of the damaged building provide some notion of the fire development that took place. Although the interest to these fire evaluations increases, it is a research field which is still in its infancy.

Except for input specification, the CFD-code itself does not provide a full physical description of the actual situation. Some incorporated models, like the turbulence models for instance, do not fully describe all aspects of turbulence since physicists and mathematicians do not know all the underlying equations. Fortunately these deficiencies in most cases are subordinate to the above described error sources.

Conclusions and recommendations
Results from the simulations demonstrate that some assumptions do affect the accuracy of results, for the better or the worse. Results in which heat and smoke are removed from the car park less efficient could result in higher risks in an actual fire situation. Since results seem to be strongly time and place dependent in some variants, these aspects should be subject to further discussion and more profound research.

The simulations themselves can be improved, for example by optimization of the User Defined Functions (especially the one describing the wall properties) or the mesh. Mesh generation based on an orthogonal grid should be preferred, probably the mesh support tools in newer versions of FLUENT (6.3 and higher, supporting polyhedral meshes) can improve the current mesh. This should be subject to further research.

When numerical data from previous simulation results obtained by TNO-CvB becomes available a more detailed comparison to these results can be achieved. In addition, the results from the simulations should be examined in higher detail to better subscribe the conclusions drawn in the previous sections.