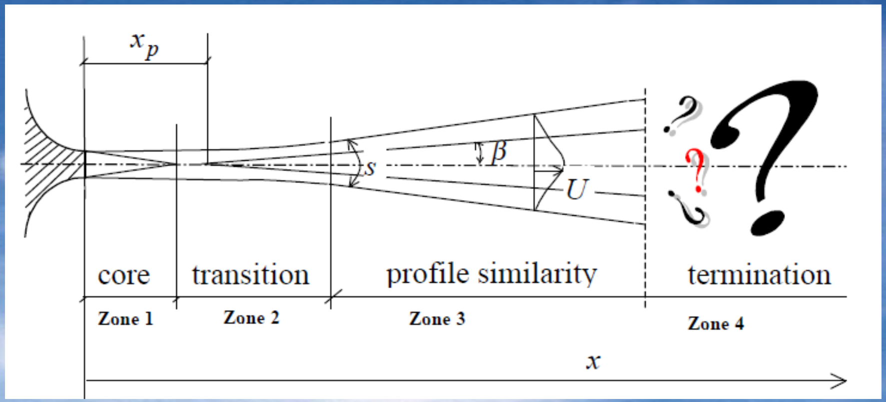
Designing a Smoke Control Car Park System in accordance with QCDD, Section 7.2



Putting theory into practice

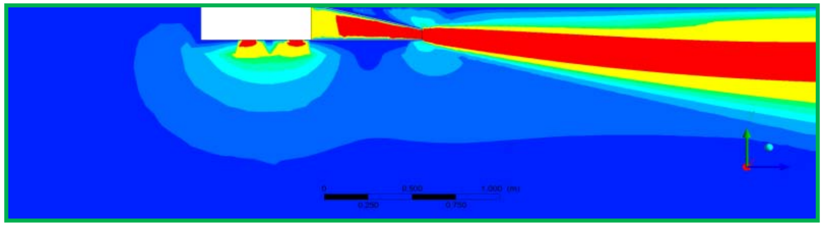
By James Allen CEng MIMechE, MCIBSE, BEng (Hons)

FläktWoods

Agenda

Part 1 – Understanding Thrust Fan capabilities

- Review of velocity profile data including CAD profiles used in our design work



- CFD vs measurement

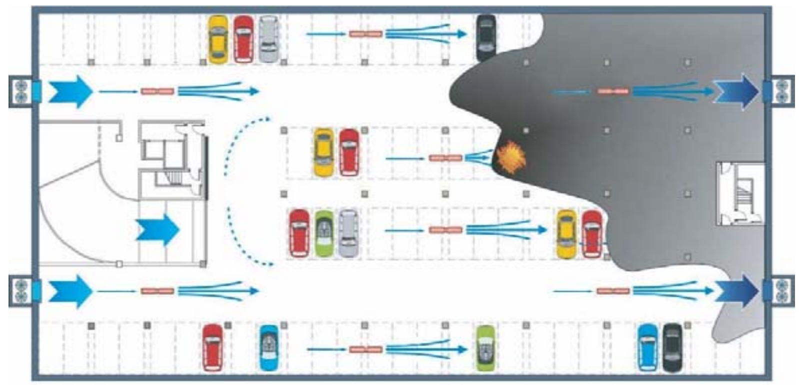
- Summary charts showing maximum area coverage per fan

- Modelling jet fans in CFD (normal flow vs component velocity vectors (radial, tangential and axial velocity)

Agenda

Part 2 - Considerations for both smoke clearance and smoke control

- Estimation of the entrainment effect influence on the extract point(s)
- Back-flow effect caused by poor fan positioning
- Effect of high inlet velocities
- Floor to ceiling height influence
- Reversibility

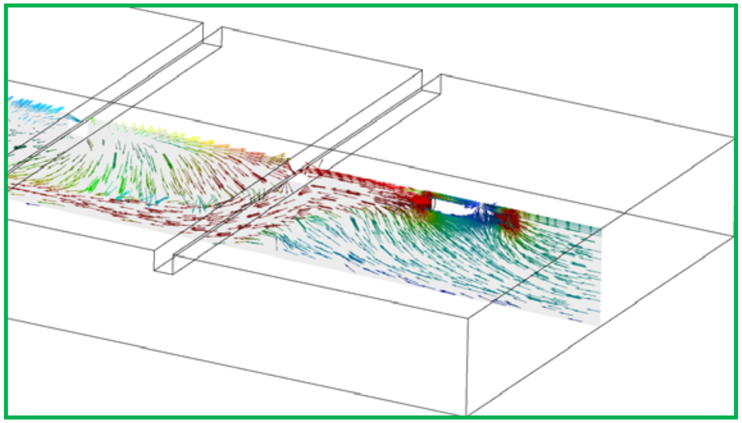


- Incorporating a sensible delay period prior to operating fans

Agenda

Part 3 – Optimal thrust fan positioning

- Wall and ceiling effects
- Installing fans in a corner



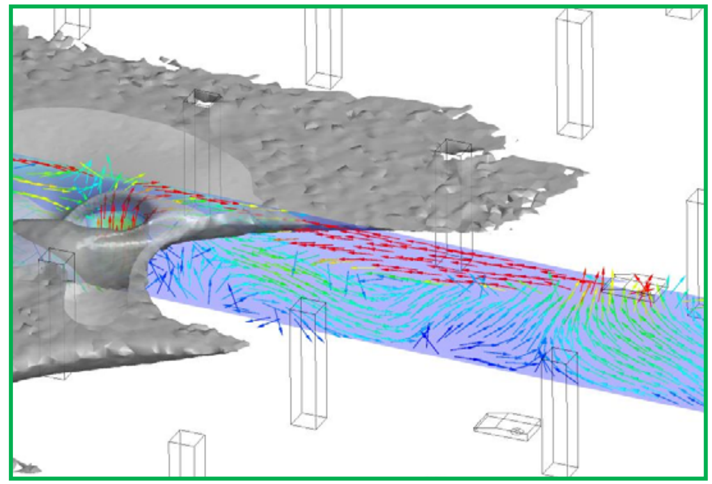
- Effect of structural pillars and down-stands
- Effect of increasing ceiling height on the jet throw profile


INTERVAL

Agenda

Part 4 – How to design for smoke control (specific to QCDD FSS-7.2)

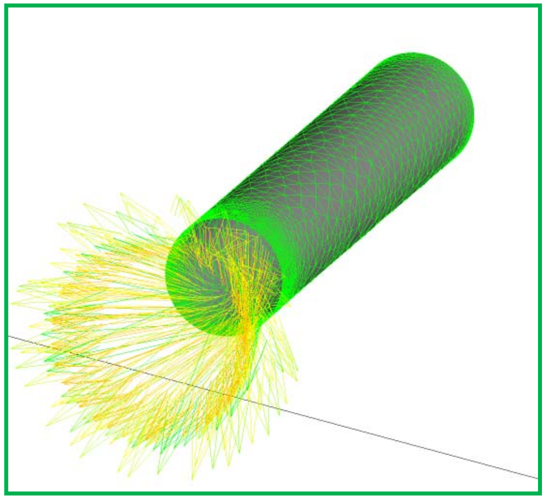
- Prediction of ceiling jet velocity of smoke from fire plume
- Smoke calculations
- Mass balance calculation
- Estimating numbers of thrust fans



Agenda

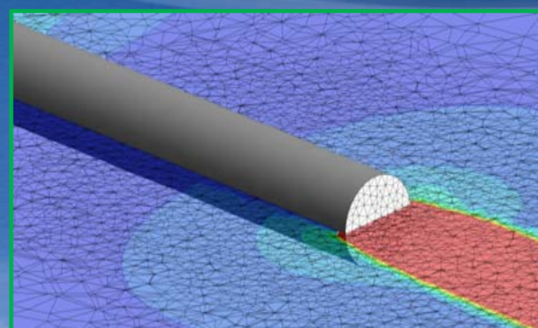
Part 5 – Use of CFD

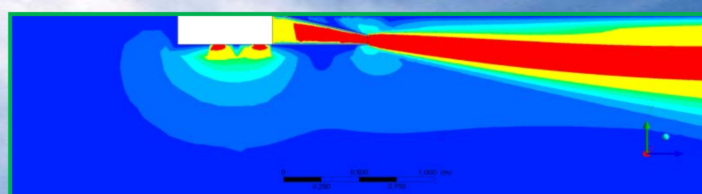
- Software types
- Importance of mesh
- Setting the correct boundary conditions for the flow
- Specifying the fire source correctly
- Convergence checks

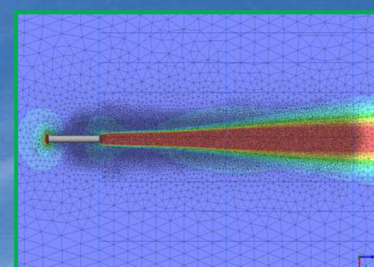


Part 1

Understanding Jet fan performance





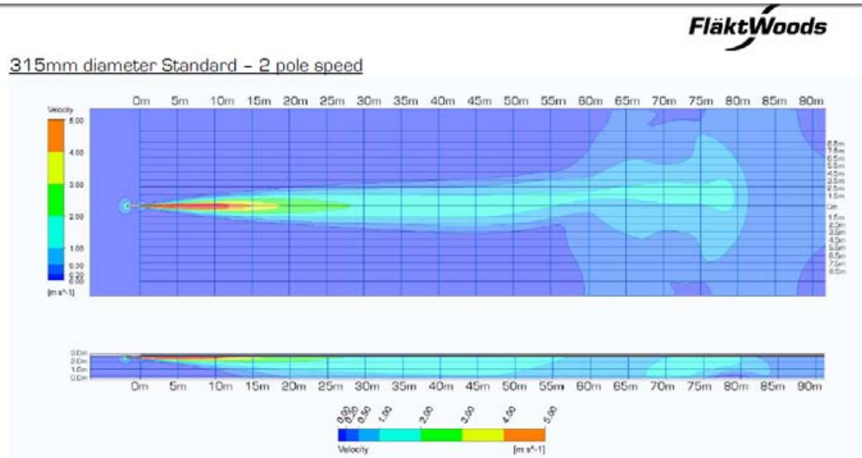


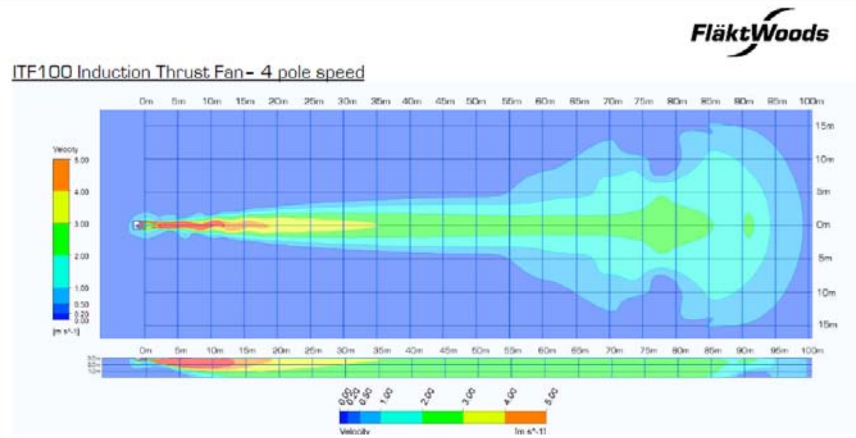
FläktWoods

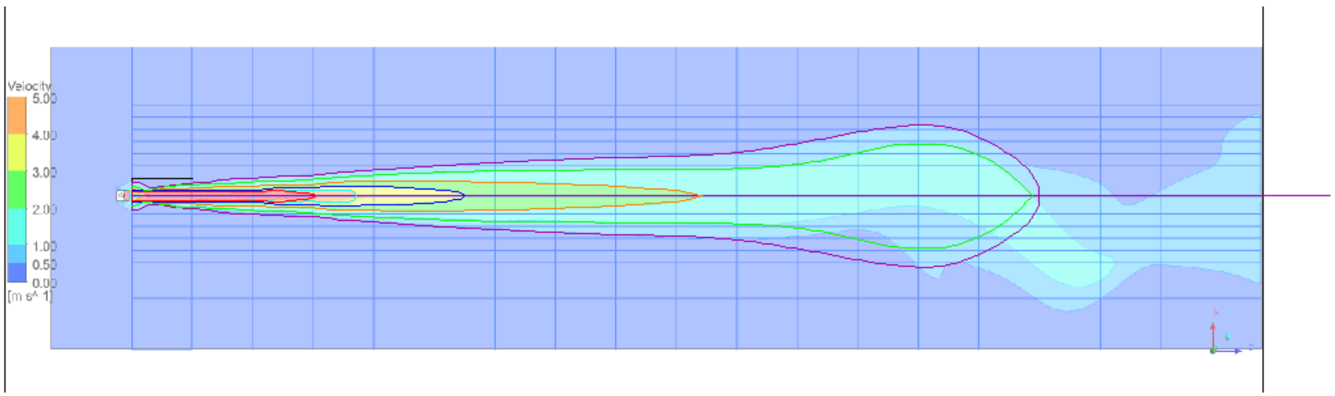
CFD modelling of Fläkt Woods range of Thrust Fans

14 profiles for Axial products

6 profiles for Induction products



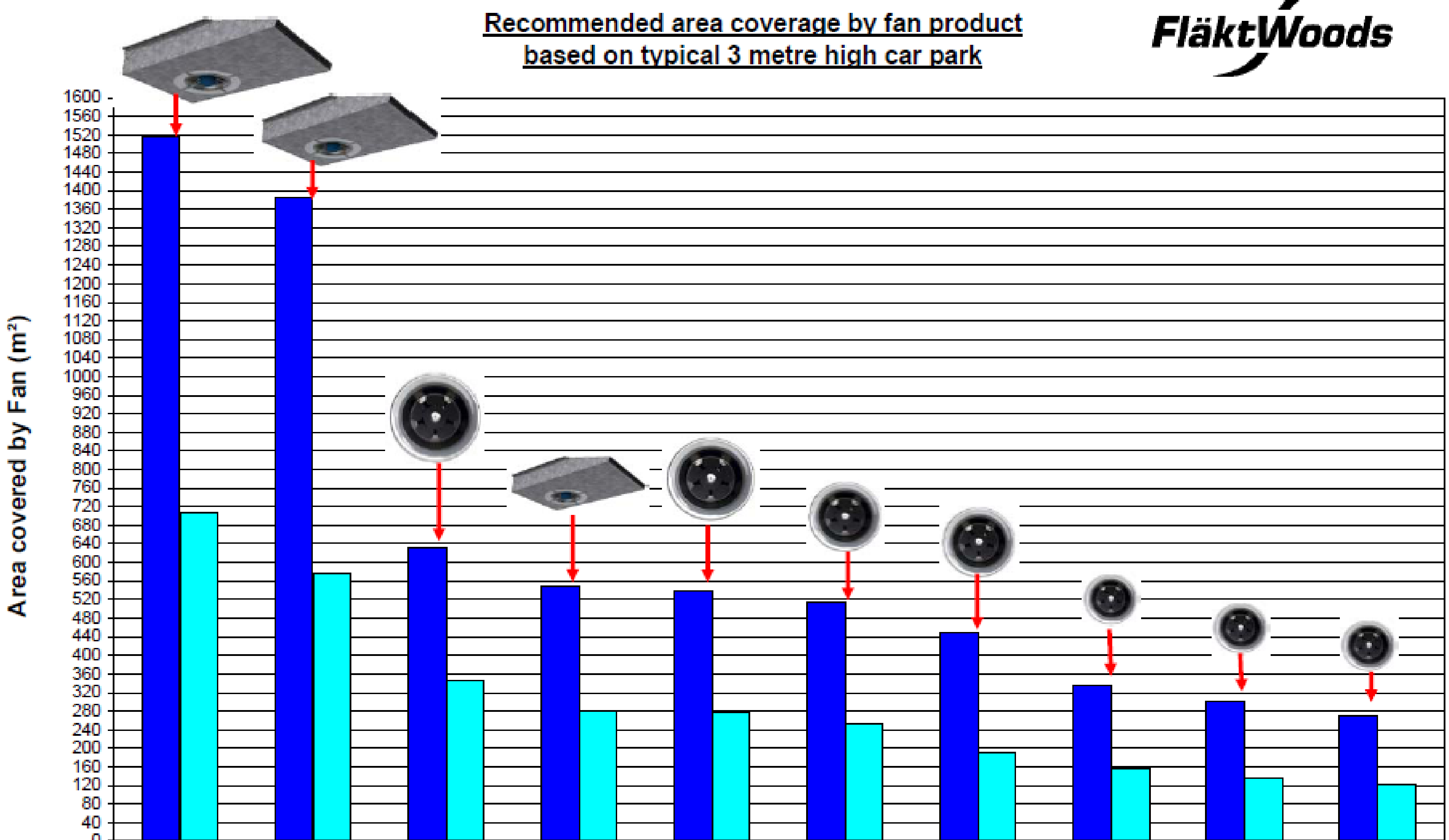




Knowledge of product performance
(every product is different, varying from supplier to supplier).

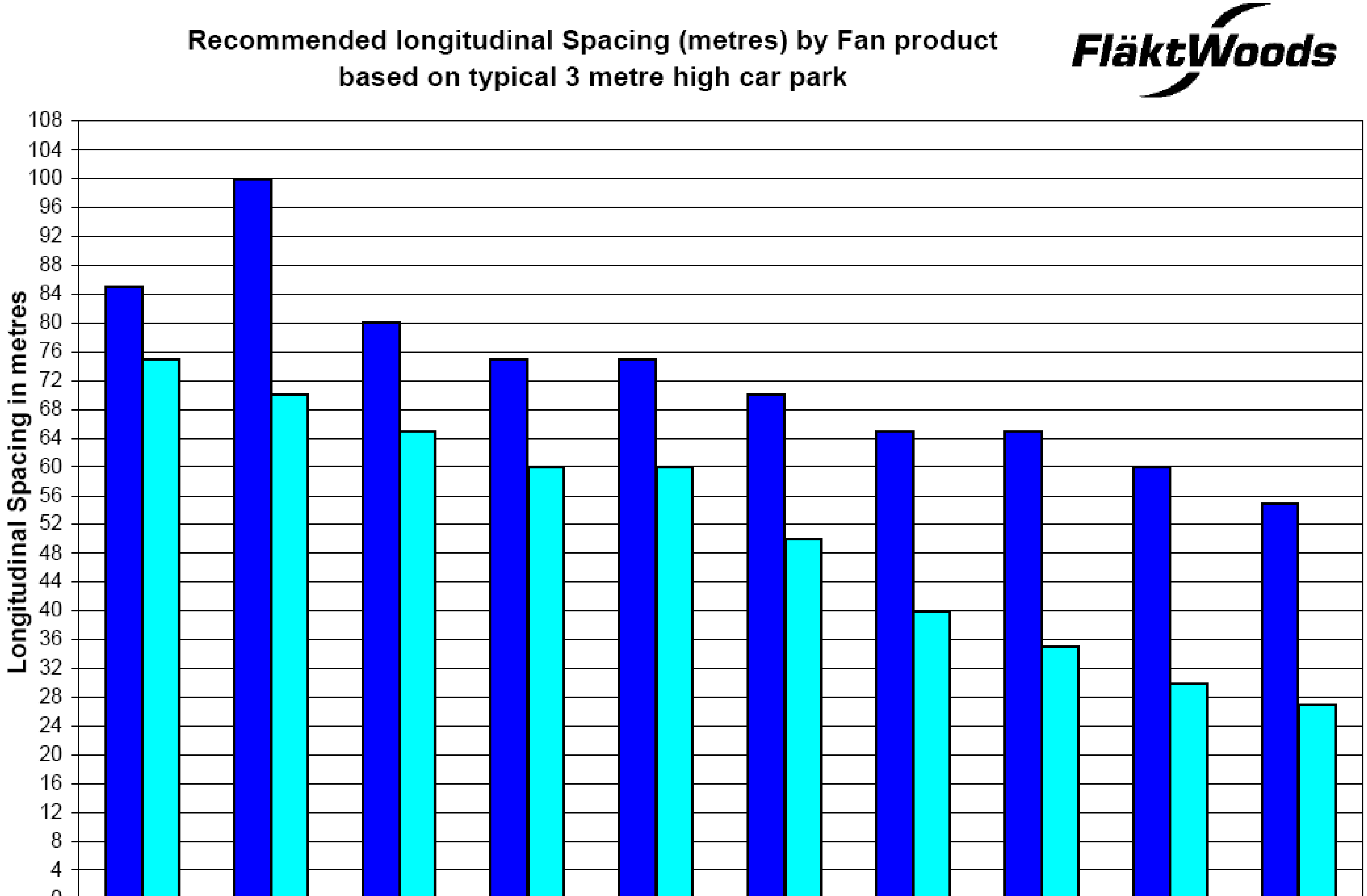
Guidance on Thrust Fan selection and positioning

Recommended area coverage by fan product based on typical 3 metre high car park



	ITF100	ITF75	40JT Max Thrust	ITF50	35JT Max Thrust	40JT	35JT	31JT Compact	31JT Max Thrust	31JT
High speed	1518	1386	633	548	537	513	448	335	303	272
Half speed	709	578	344	281	276	252	189	158	136	121

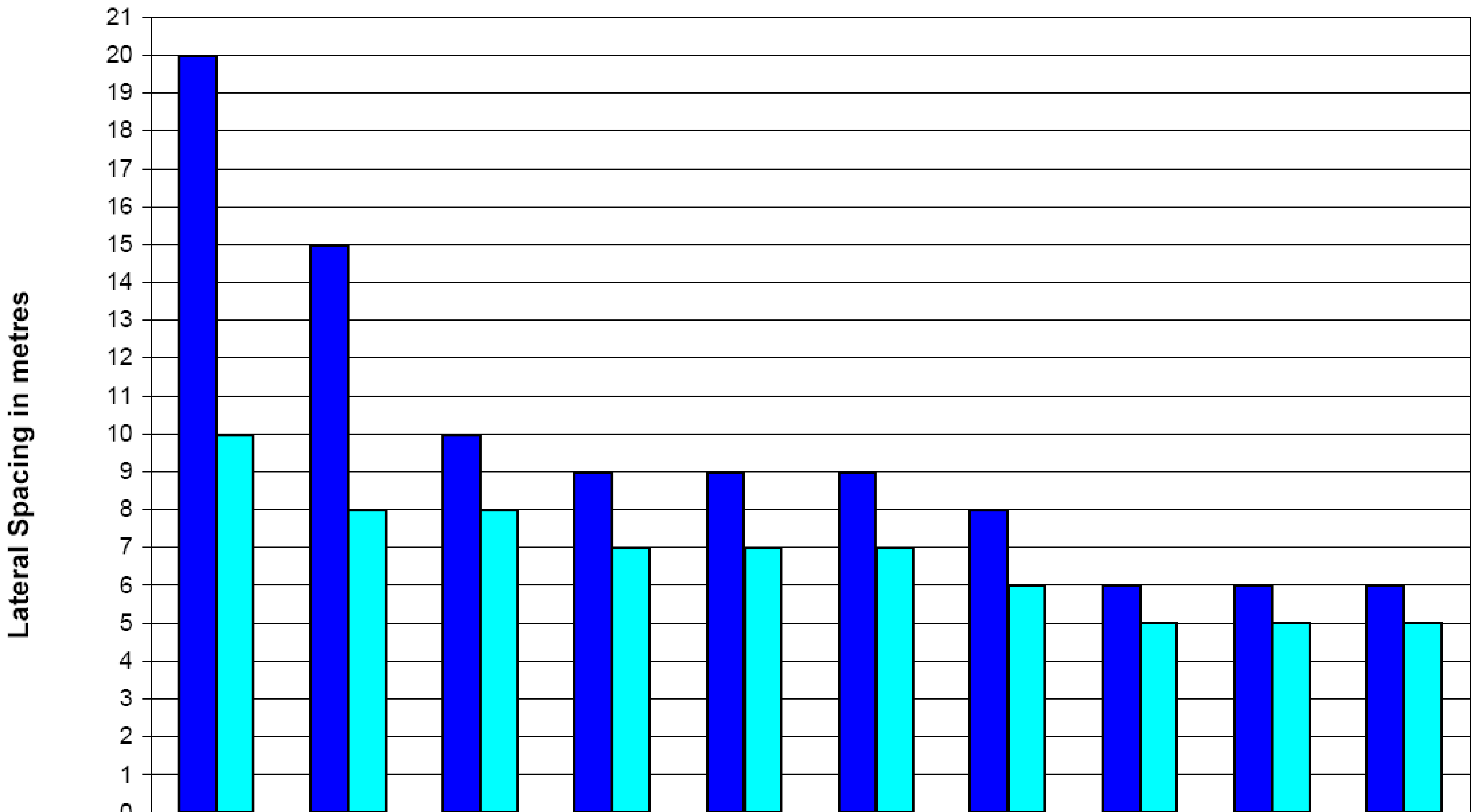
Guidance on Thrust Fan selection and positioning



	ITF100	ITF75	40JT Max Thrust	ITF50	35JT Max Thrust	40JT	35JT	31JT Compact	31JT Max Thrust	31JT
High Speed (m)	85	100	80	75	75	70	65	65	60	55
Half Speed (m)	75	70	65	60	60	50	40	35	30	27

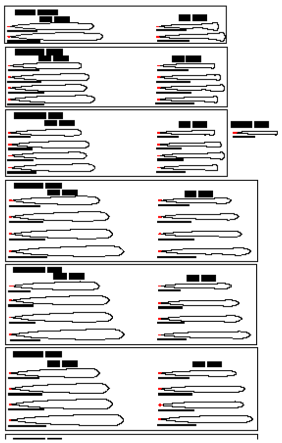
Guidance on Thrust Fan selection and positioning

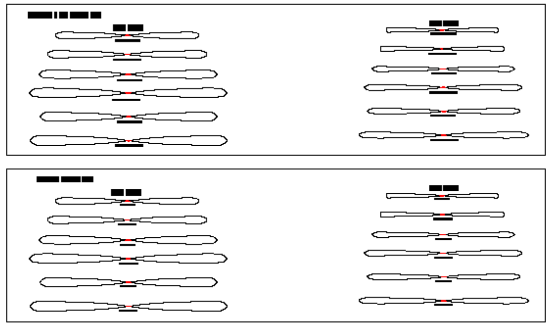
Recommended lateral Spacing (metres) by Fan product
based on typical 3 metre high car park

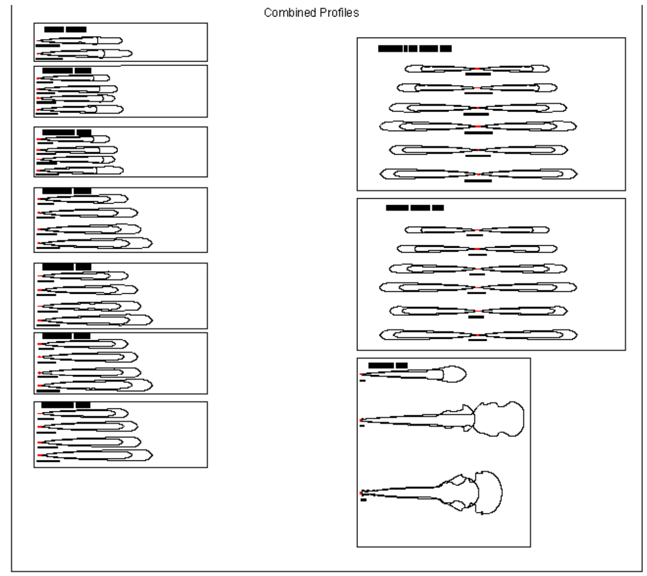


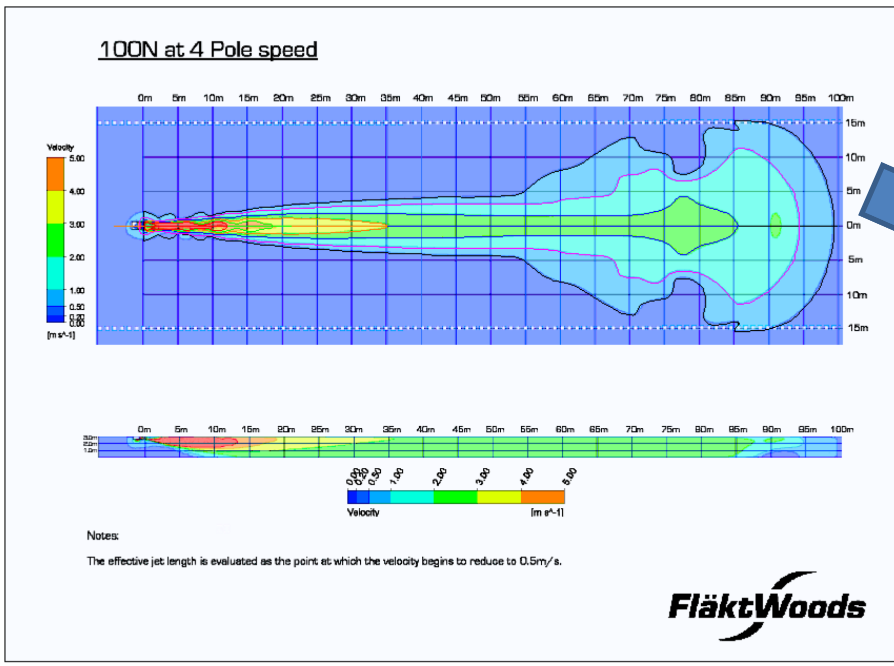
	ITF100	ITF75	40JT Max Thrust	ITF50	35JT Max Thrust	40JT	35JT	31JT Compact	31JT Max Thrust	31JT
High speed (m)	20	15	10	9	9	9	8	6	6	6
Half speed (m)	10	8	8	7	7	7	6	5	5	5

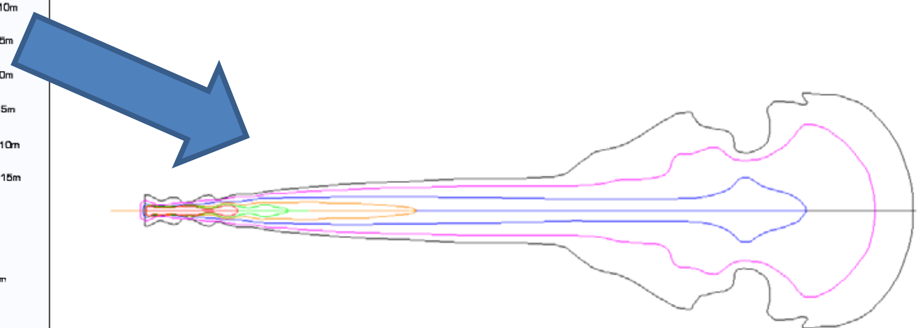
CFD modelling translated into CAD



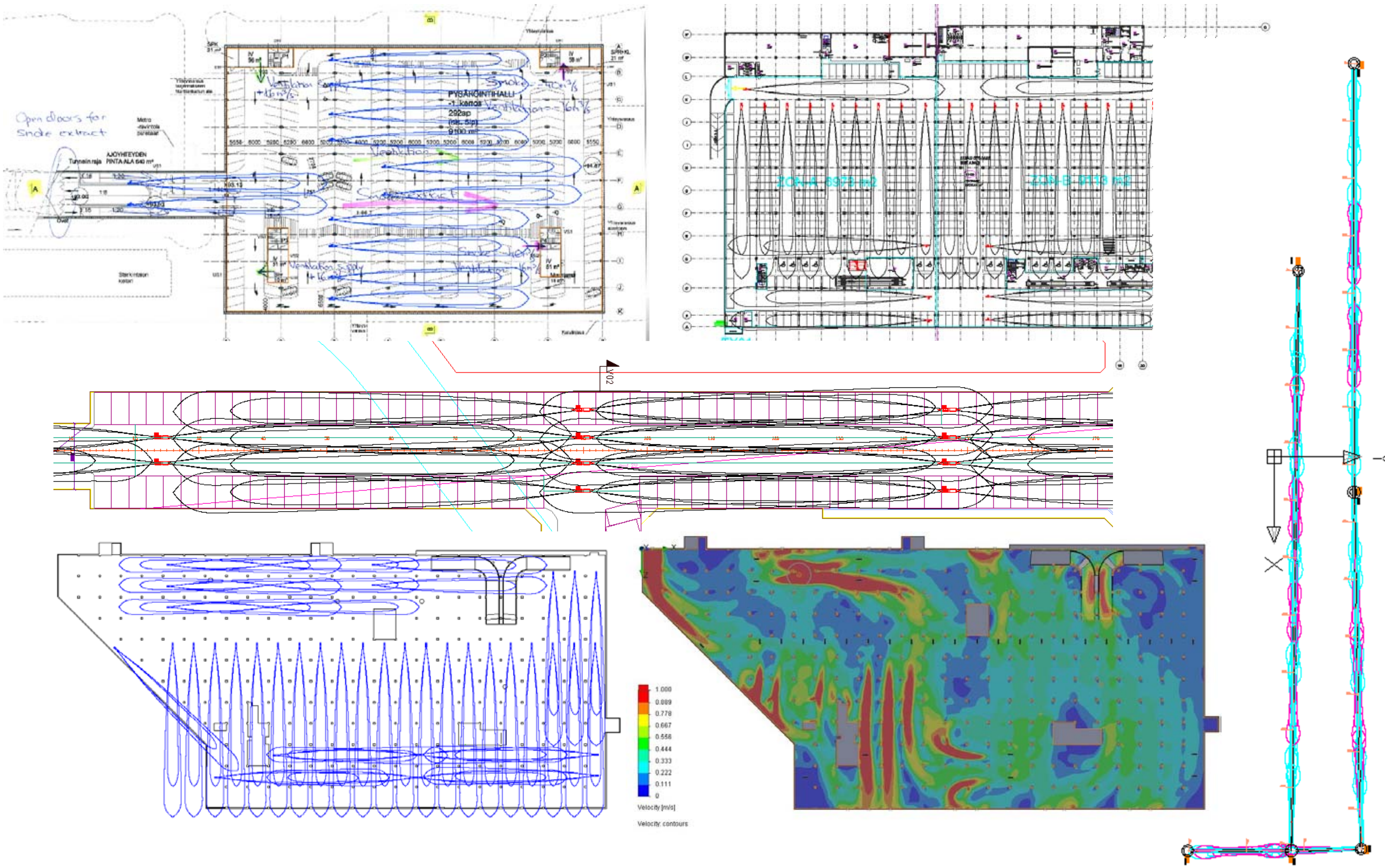








Examples of CAD profiles in use



Measurement vs CFD

VENUE AT SITE: Singapore Expo, 1 Expo Drive, S'486150

TEST SAMPLE: 'Fläkt Woods' ductless jet thrust fan
Model: HT35JM.JET/4/SP 1L 80Z
Serial number: 21672/01A
Date of manufacture: 06/06
(Made in England)
Length: 1800 mm
Nozzle outlet : Ø355mm
Power Comsumption: 180 W
Speed: 1420 RPM
Power Supply: 1ph/220~240V/50Hz
Capacitor: 20 uF
Current: 1.35 A

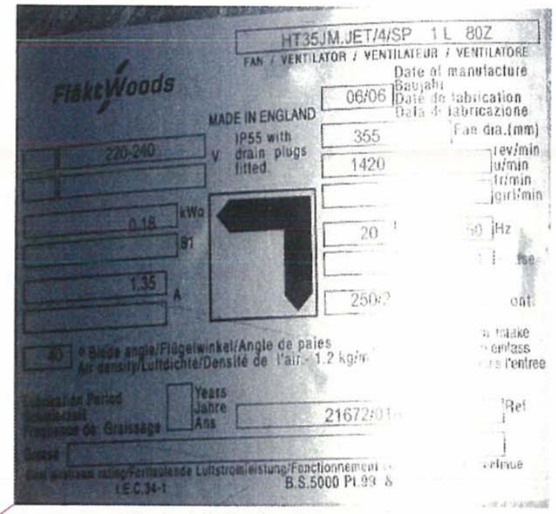


Photo 1: Label on test specimen

Photo 2 : Fan installation in Singapore EXPO (Front view)

Photo 3 : Back view

Measurement vs CFD

TEST SETUP:

The EXPO hall space is approximately 100m(L) X 50m(W) X 12m(H), had been sourced for the 'Air velocity distribution' test. The 'Fläkt Woods' ductless jet thrust fan was installed and levelled horizontally at a height of 3m above the floor level, located at one end of the floor space. Grid-lines were marked on the floor with masking tapes for the air-velocity distribution data collection. Air-velocity meter was setup rigidly and able to move around easily during the test.

Photo 4: Front view of fan installation

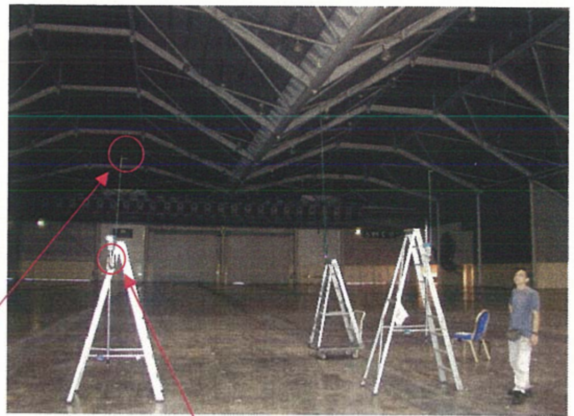


Photo 5: Air-velocity meter setup

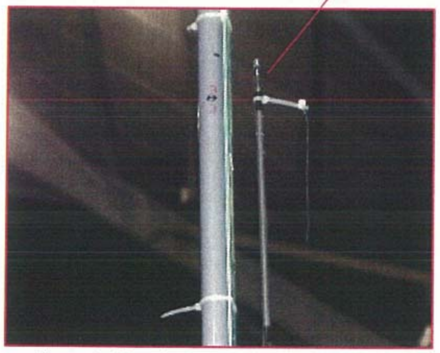
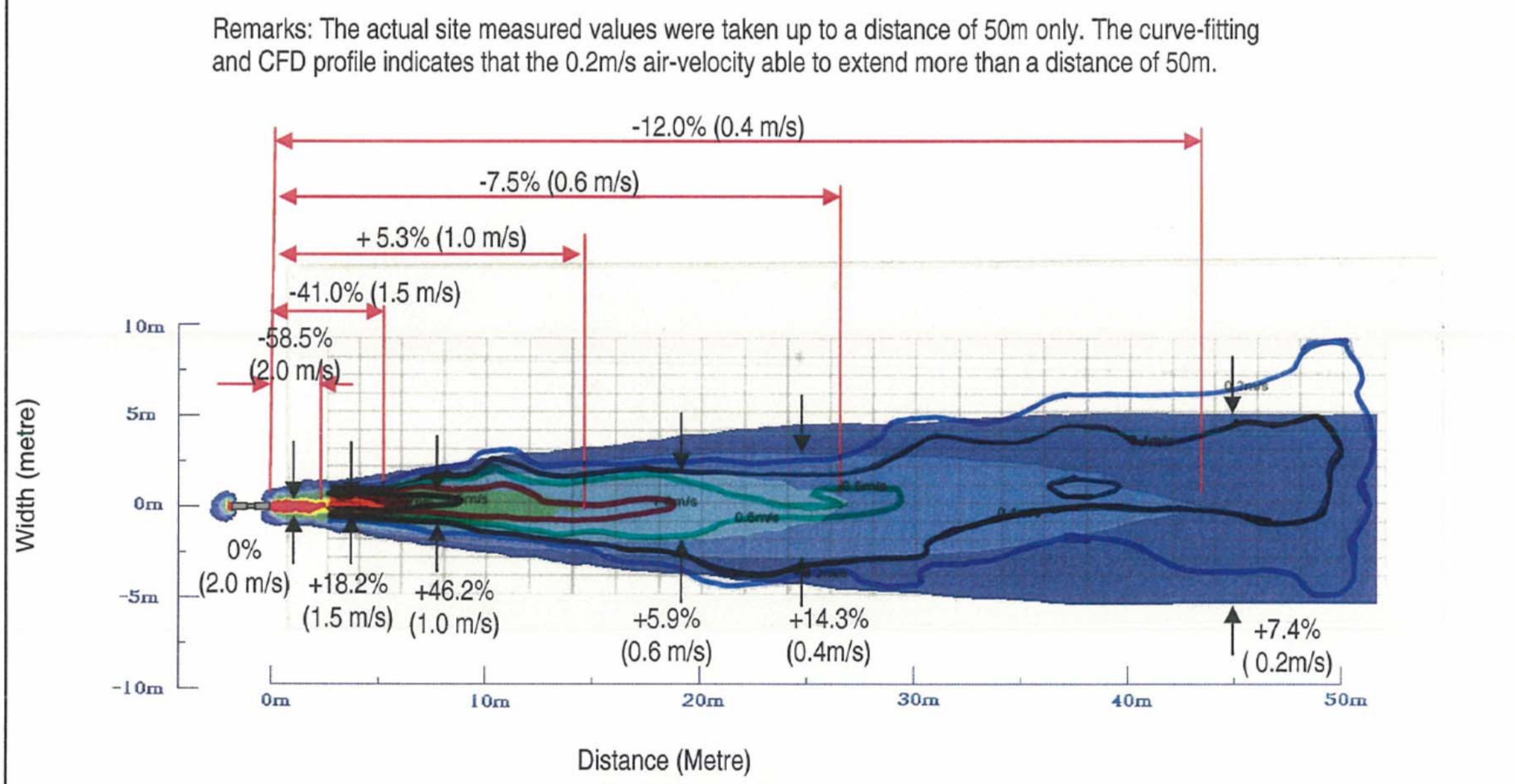


Photo 6: Air-velocity meter measuring probe

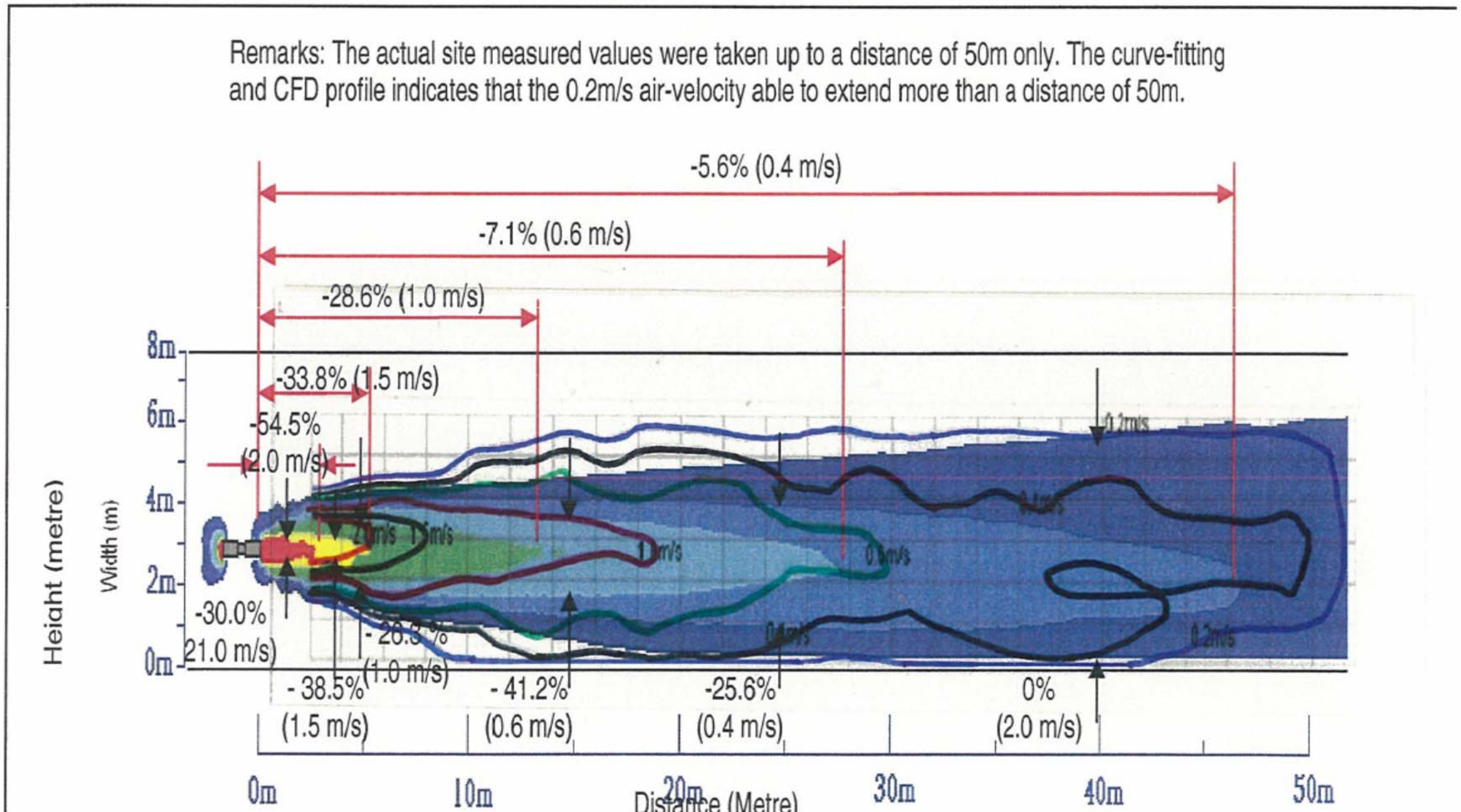
Photo 7: Air-velocity meter

Measurement vs CFD – Plan view



PLAN VIEW – CFD Simulation

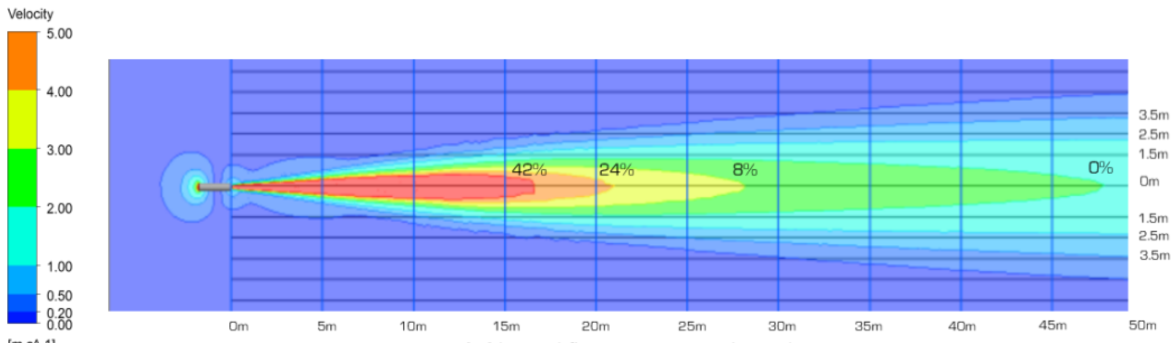
Measurement vs CFD – Side view

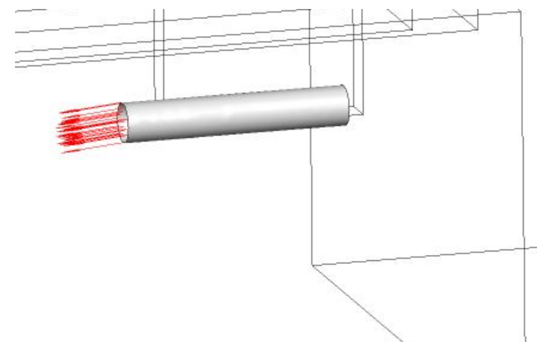


SIDE VIEW – CFD Simulation

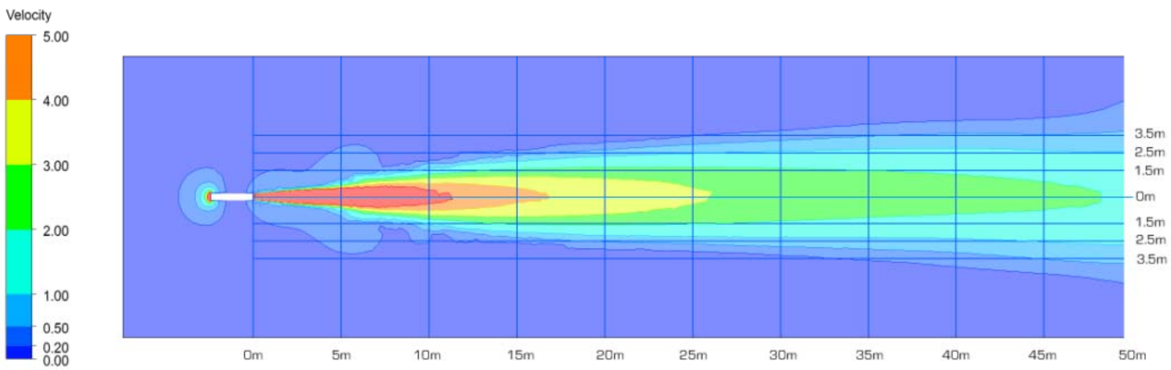
Modelling fans - Normal flow component vs velocity components

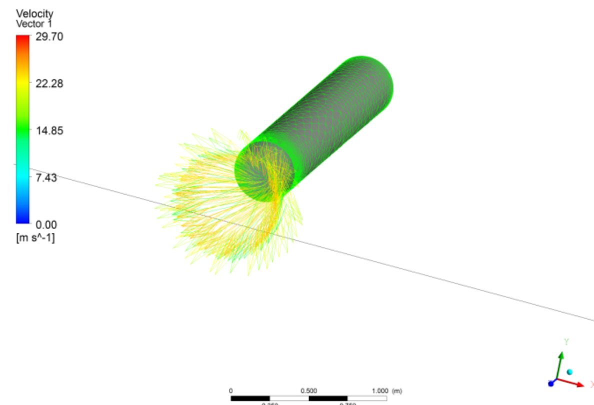
Comparison of Normal flow component boundary condition with axial, tangential and radial velocity components





A: Normal flow component boundary



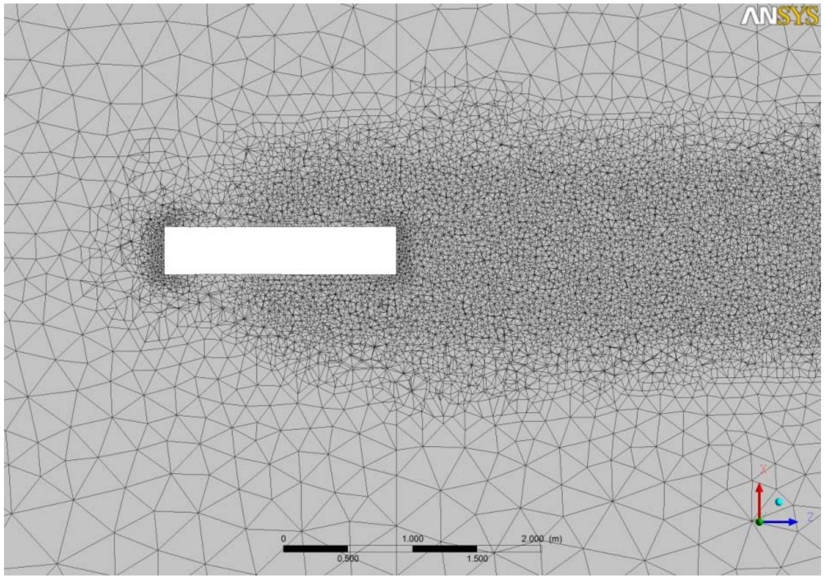


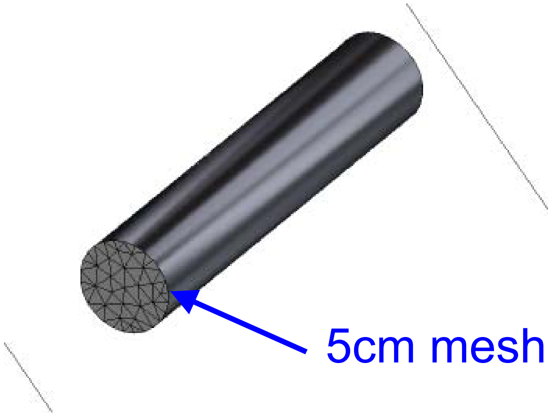
B: Axial, radial and tangential velocity components based on CFX simulation of airflow across the fan blades

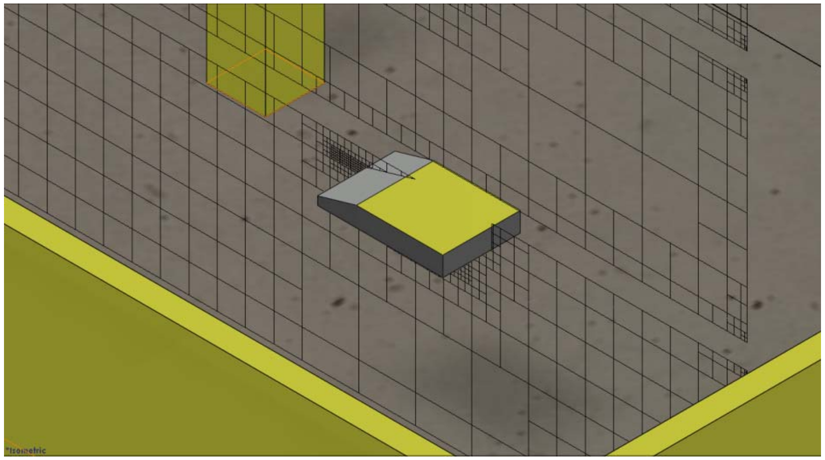
Notes:

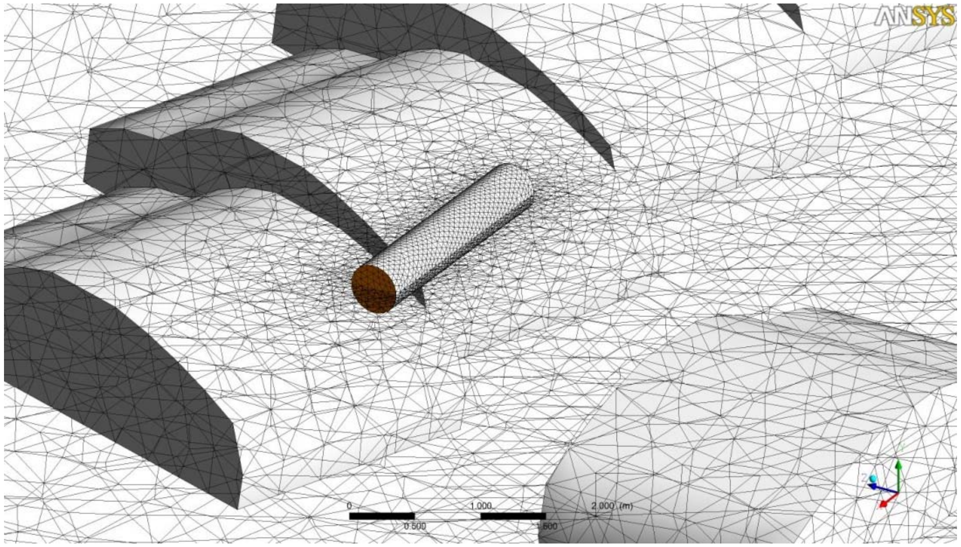
Deviation = [(A - B) / B] x 100%

Modelling Thrust Fans in CFD – Choice of mesh

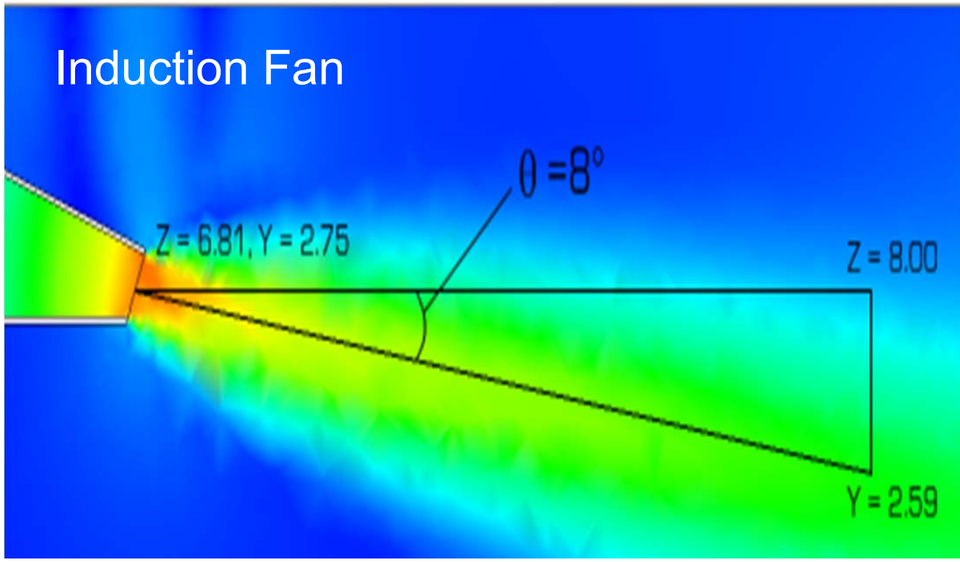


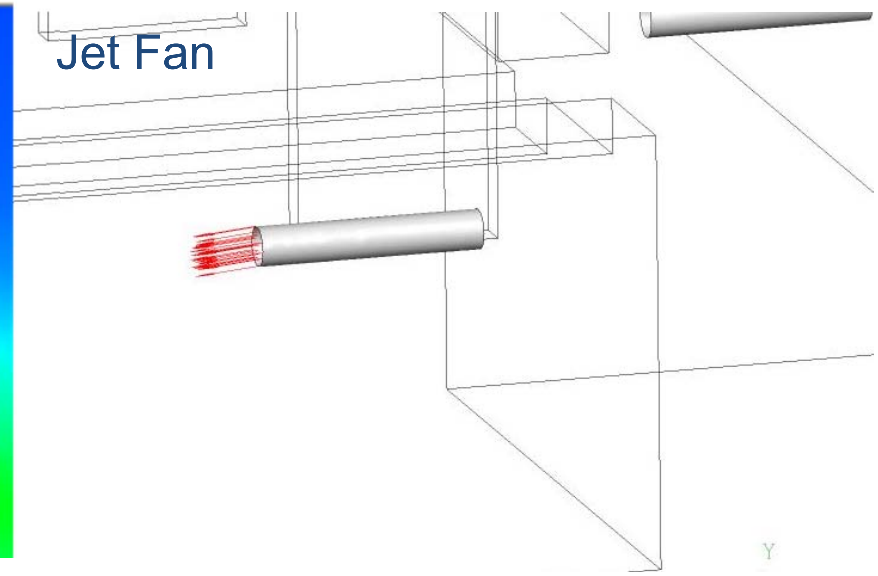






Jet flow angle for Flakt Woods products





ITF100 = 8°

ITF75 = 4°

ITF50 = 6°

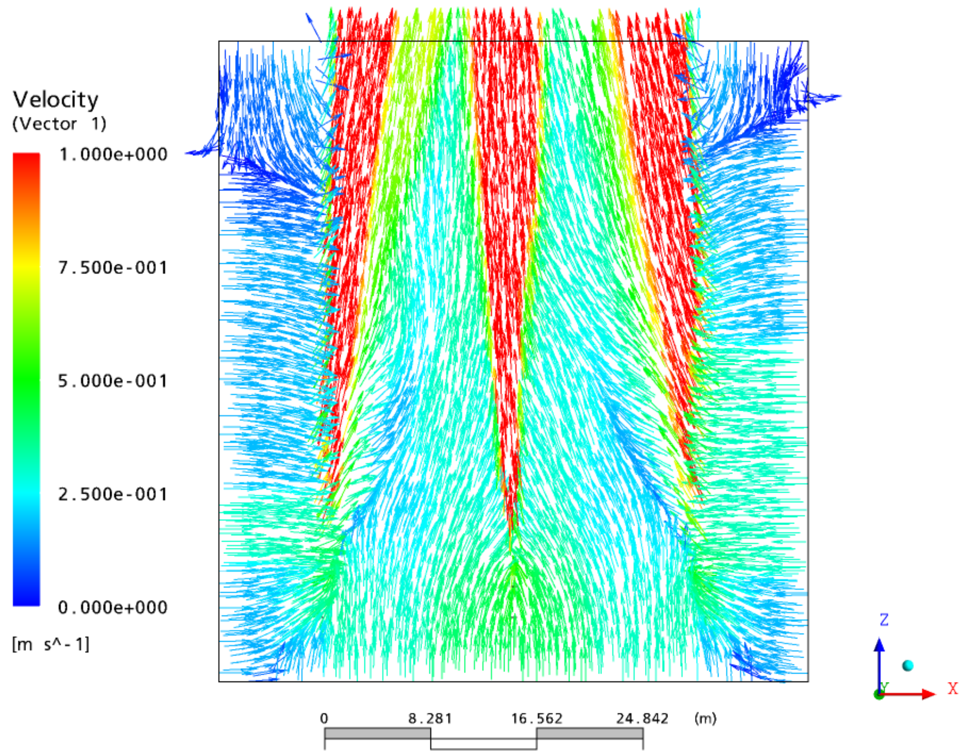
Axial fans where beams are present optimum deflector angle = 5°

Flow angle of the jet from each fan product is critical to performance and position in relation to fixed objects (beams etc)

Part 2

Considerations for smoke clearance and smoke control

Importance of entrainment ratio



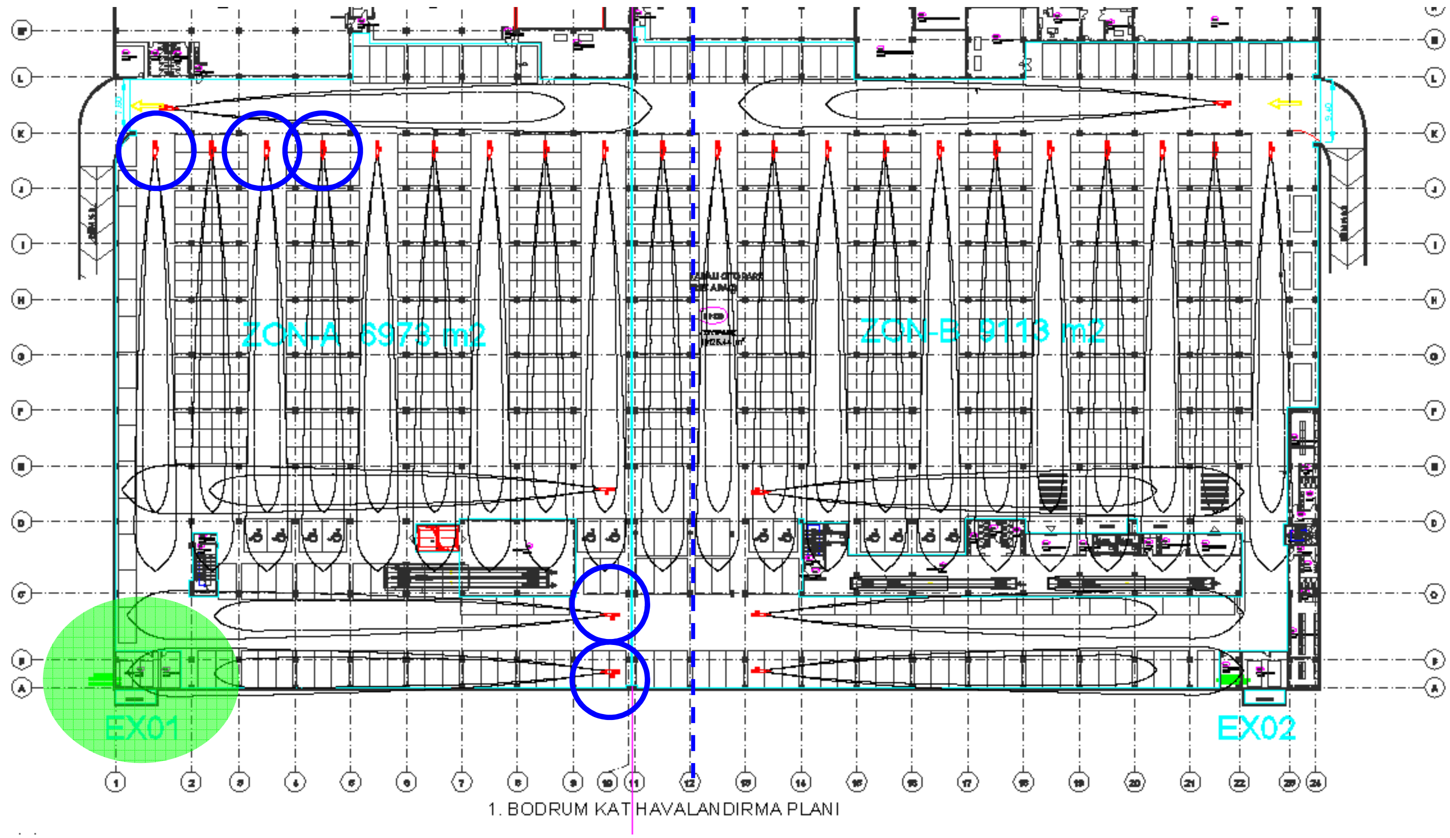
Total flow rate = flow through the fan + entrainment

Axial fans = 4 to 6 times flow rate through the fan

Induction fans = 8 to 9 times the flow rate through the fan

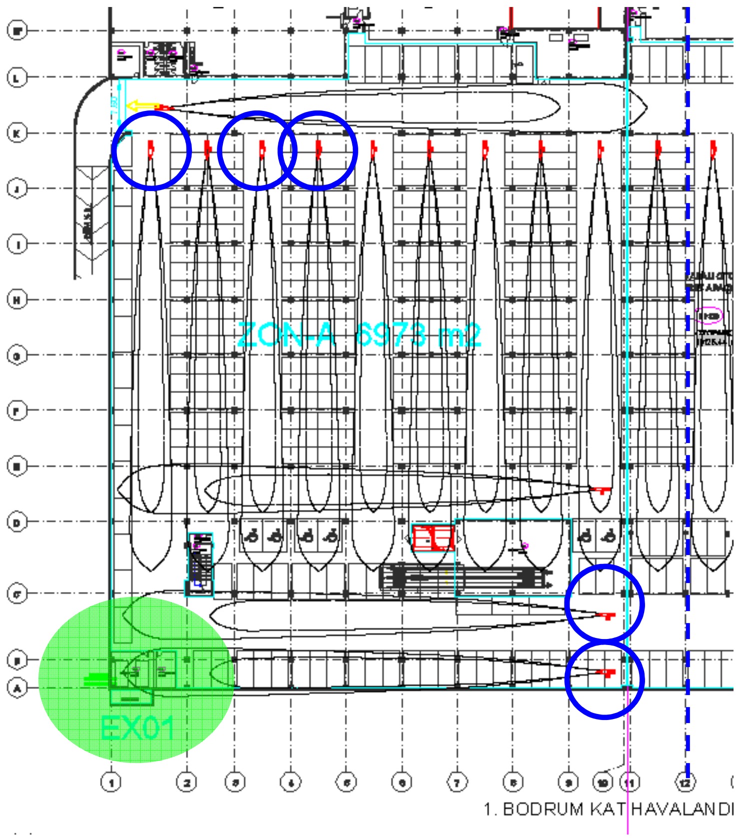
Importance of balancing Thrust fan and extract flows

Thrust fans have two main functions – mixing & accelerating air towards the extract



Importance of balancing Thrust fan and extract flows

Thrust fans have two main functions – mixing & accelerating air towards the extract



In this example 5 fans are directed towards extract. Calculate total induced flow that these fans provide and check that this does not exceed the extract flow rate.

Consider an installation factor in your calculations i.e.

Smooth ceiling = 0.8 to 0.9

Obstructions in front of fan(s)

= 0.3 to 0.6 (dependant on spacing and depth)

Installation factor is only applicable to Thrust fans

Velocity effect

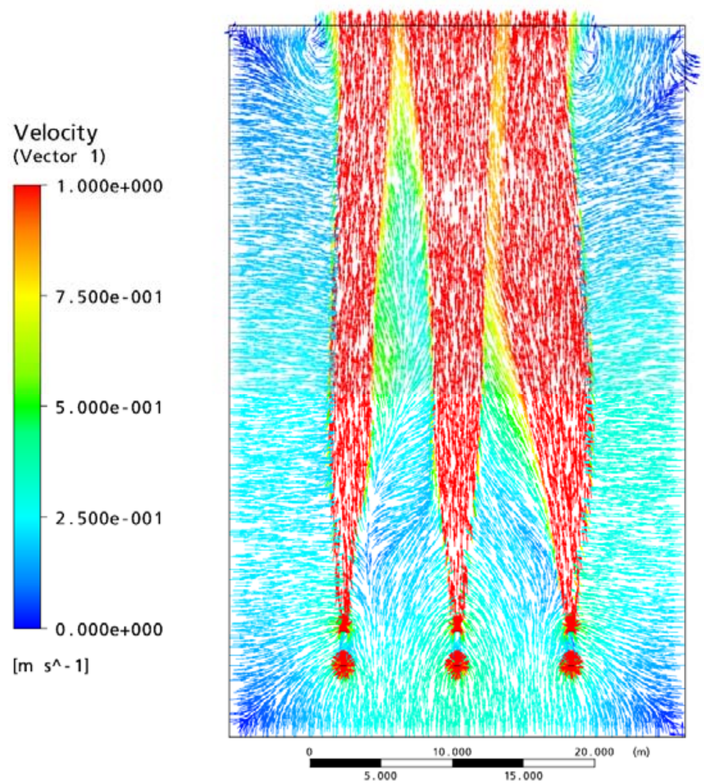
Velocity effect = 1 – Vc / Vf

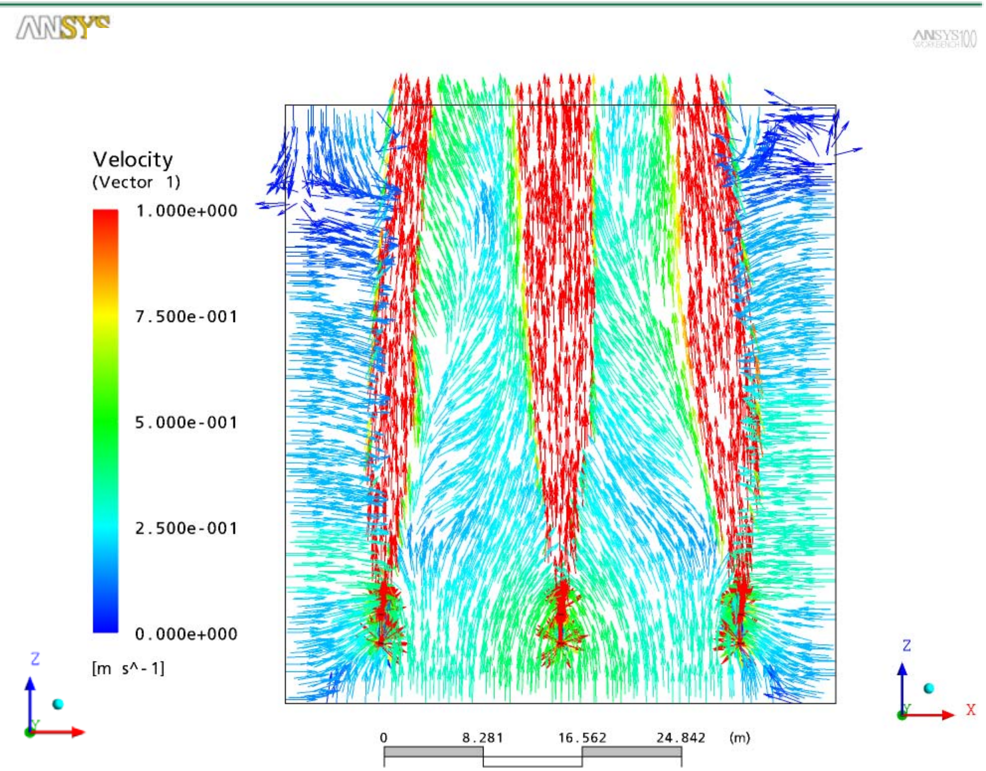
Where Vc = Velocity induced by extract (average over car park cross-sectional area)

Vf = Velocity at the outlet of the Jet fan

Design velocity = required velocity / (Installation effect x velocity effect)

Importance of positioning fans at the correct spacing



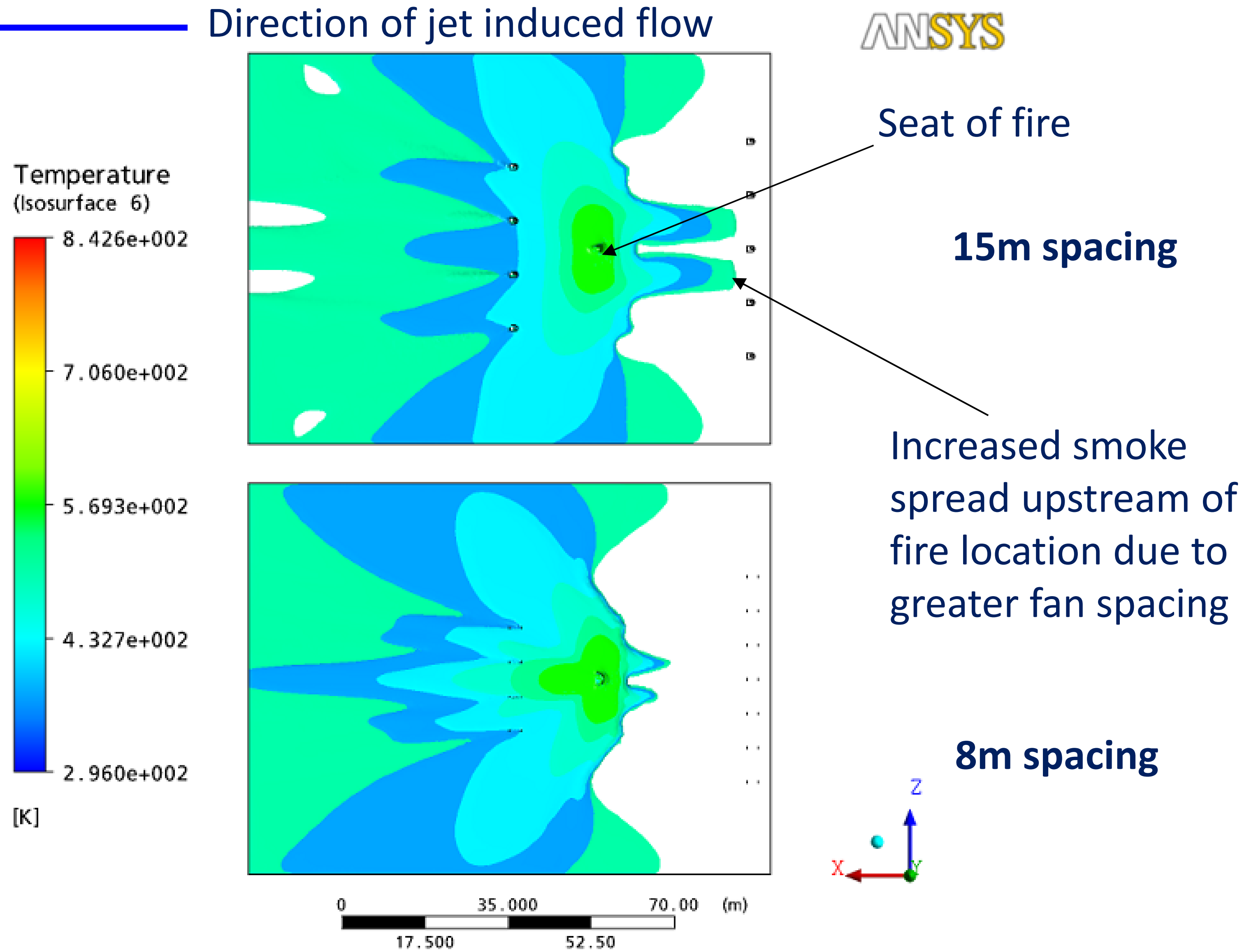


Increased spacing between Thrust Fans (>recommended limits) can mean higher extraction rates are required for smoke control.

Velocity effect (1 – Vc / Vf) is reduced

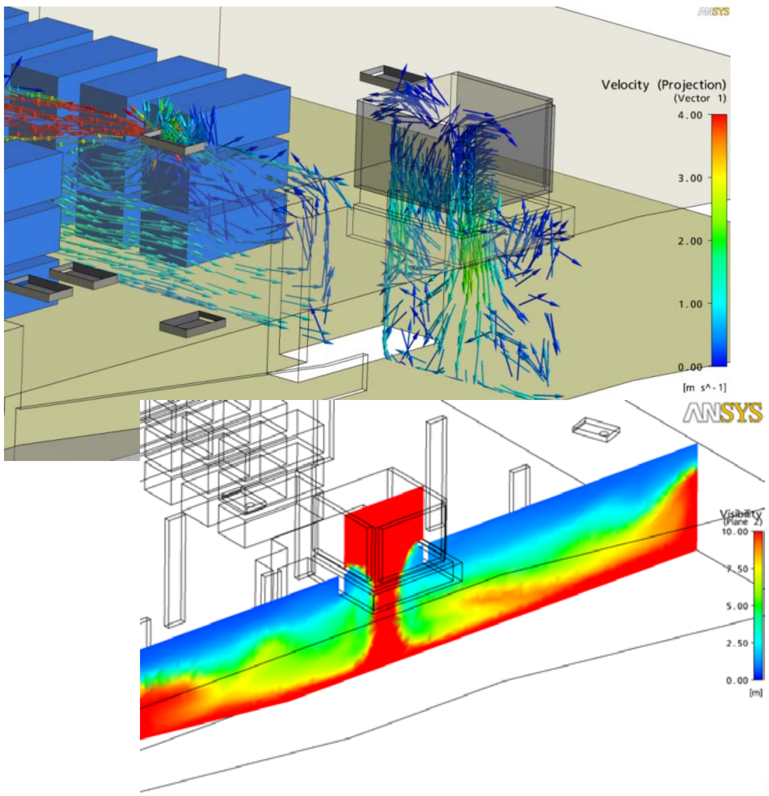
Importance of positioning fans at the correct spacing

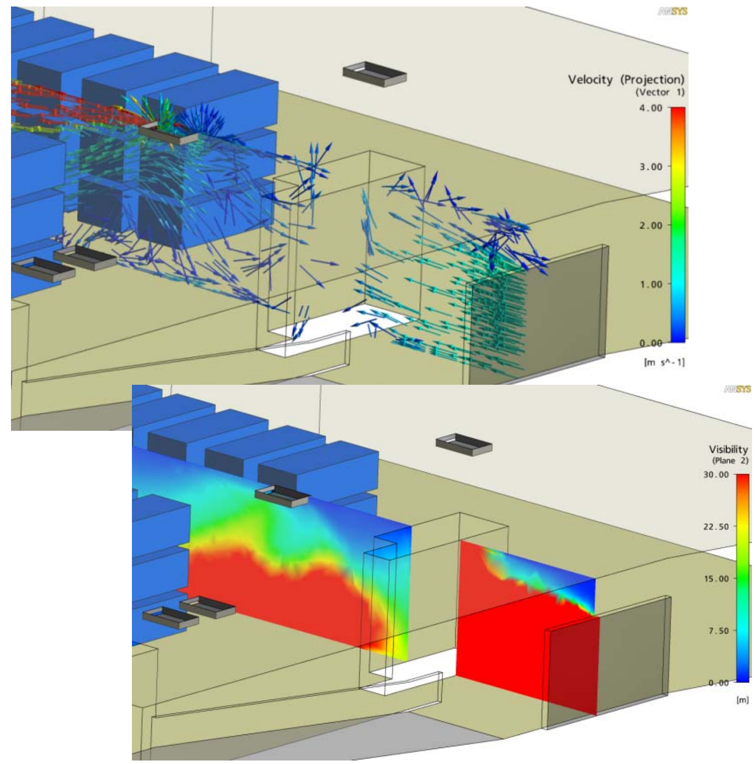
Direction of jet induced flow



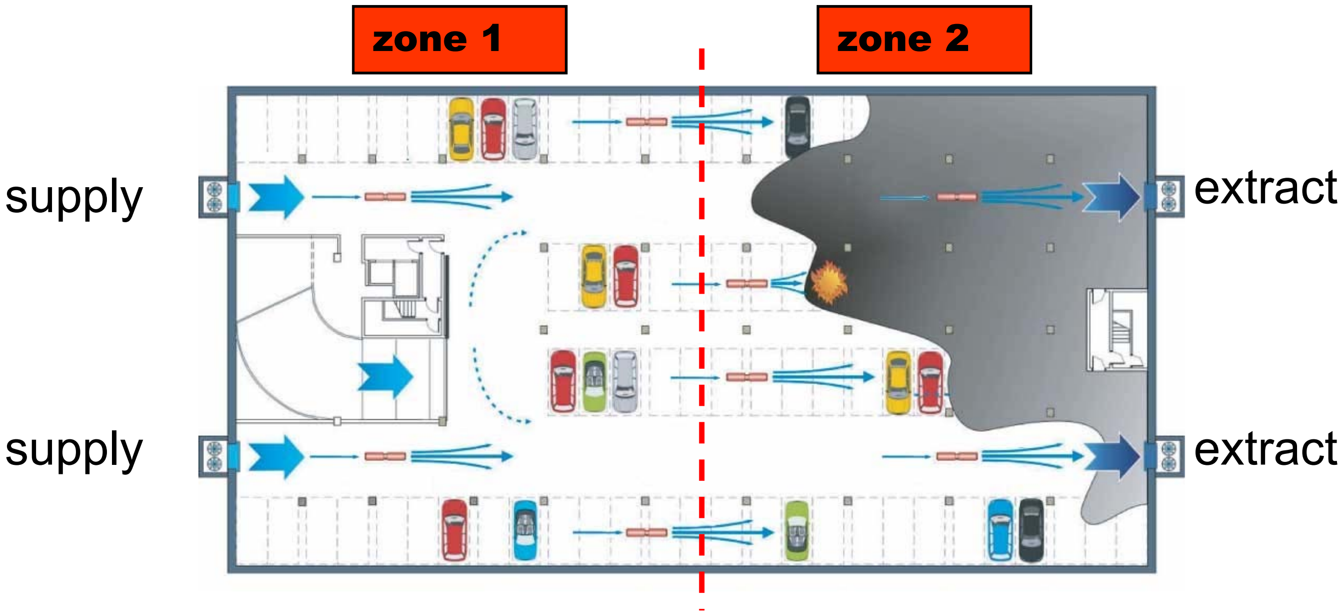
Effect of high inlet velocities

- Too high inlet velocities can caused unwanted recirculation / backflow.
- **Ideal inlet velocity is between 1 to 2m/s.**
- Higher inlet velocities can be designed for but need to be verified carefully using CFD modelling.
- Position of inlets are also important.





Reversibility

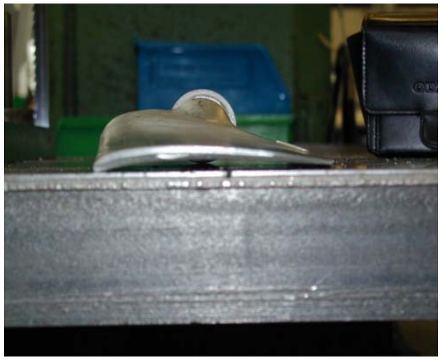


Unidirectional: 50 to 60% flow in reverse. Not suitable for continuous operation (ok for one off)

Truly symmetrical:

100% in reverse

Suitable for continuous operation



Importance of delaying fan operation

Clear height maintained during evacuation.

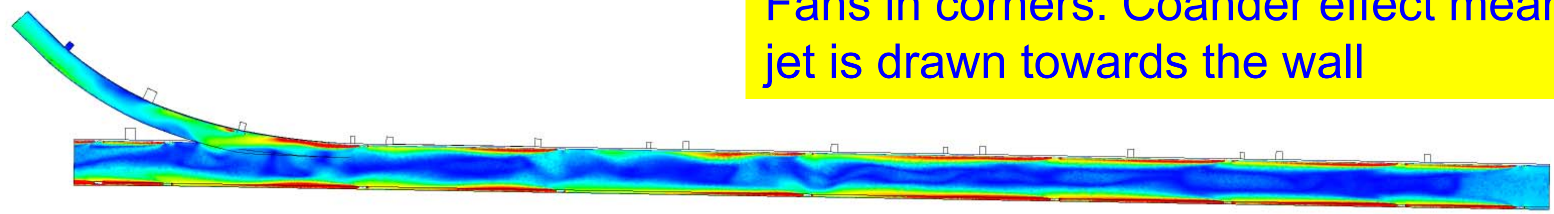
Delay period should be set as time taken for all occupants to evacuate (specific to each project)

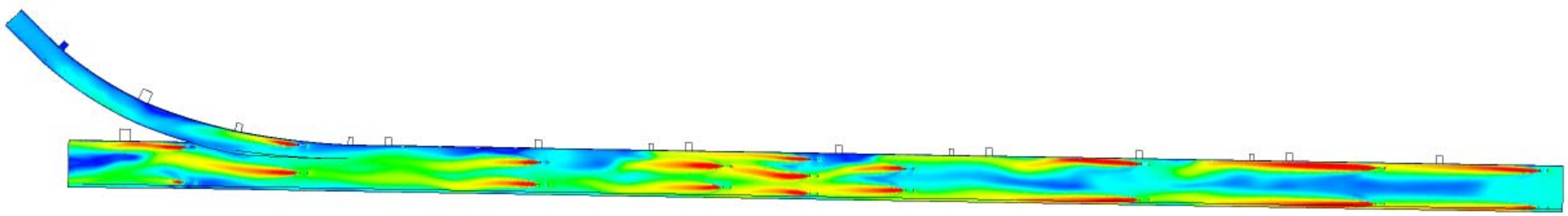
Fogging effect downstream of Thrust Fans due to break up of smoke layer

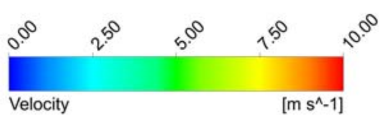
Part 3
Optimal Thrust fan positioning

Wall and ceiling effects

Fans in corners. Coander effect means the jet is drawn towards the wall





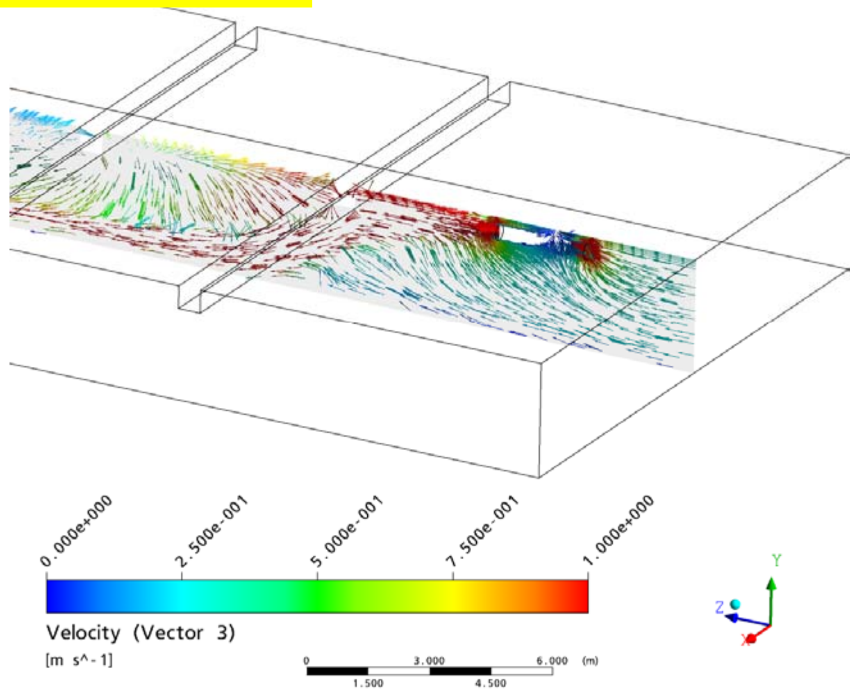


Fans moved further away from tunnel wall the tunnel velocity increases.

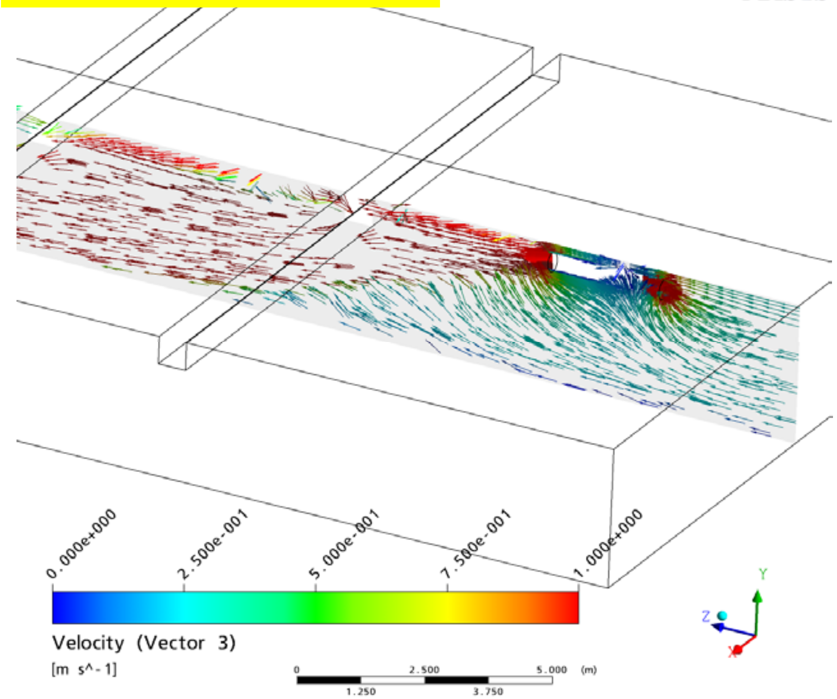
Velocity effect 1-Vt / Vf is higher

Beam effects

No deflector

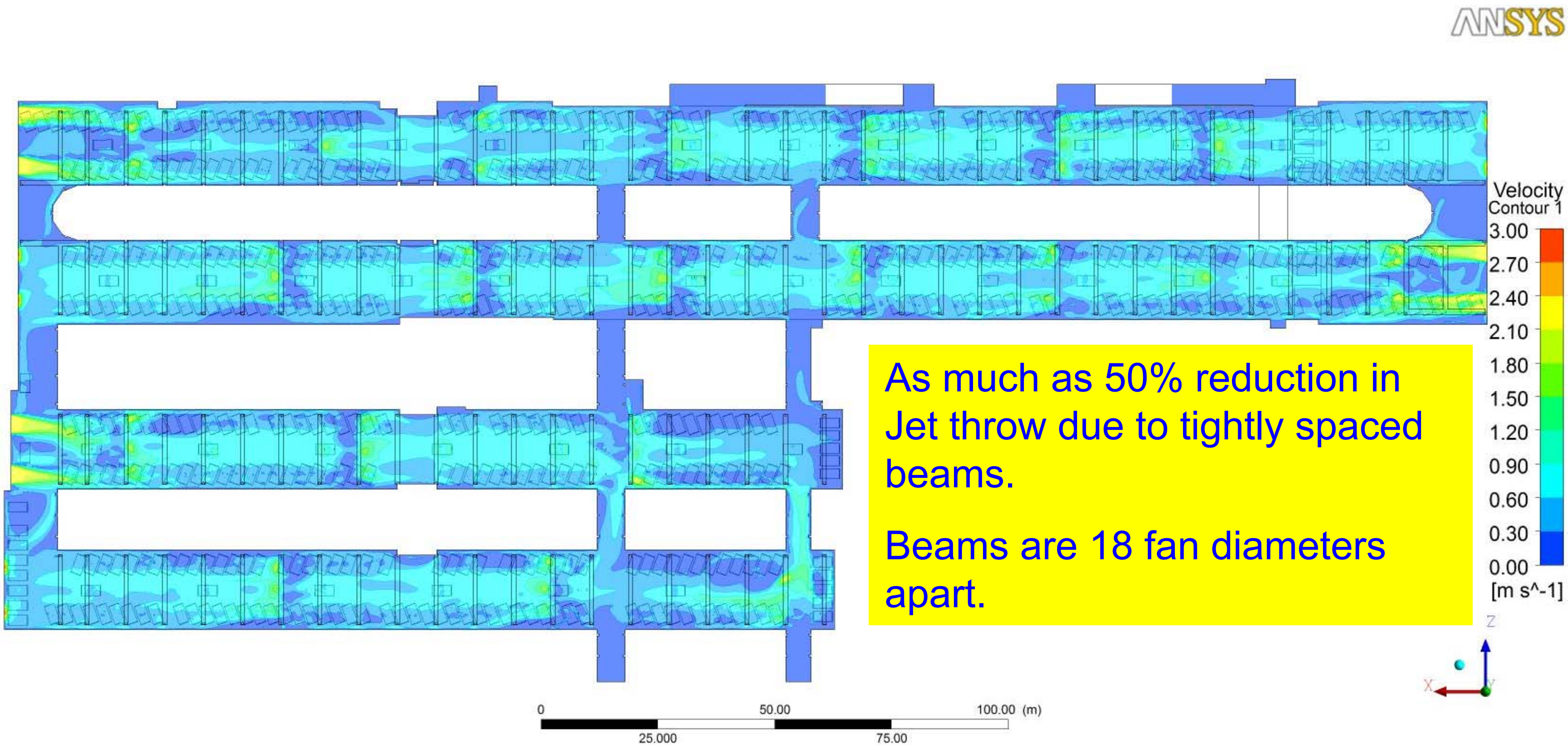


Deflector fitted



Spacing jet fans at >10 fan diameters with 5° deflection angle will minimise effect of beam

Beam effects

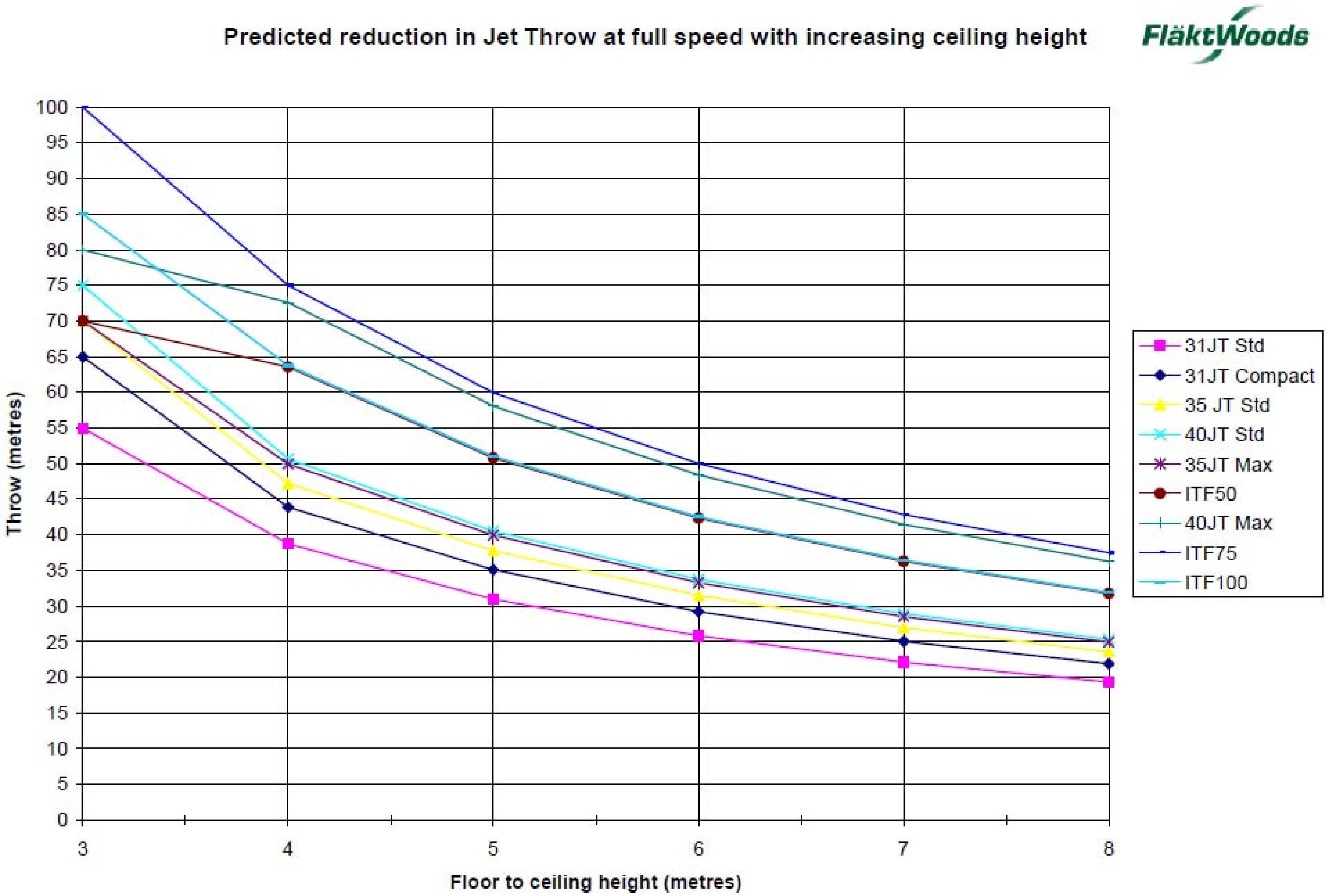


As much as 50% reduction in Jet throw due to tightly spaced beams.

Beams are 18 fan diameters apart.

In this scenario use an installation factor of approx 0.5 when preparing hand calculations

Effect of floor to ceiling height

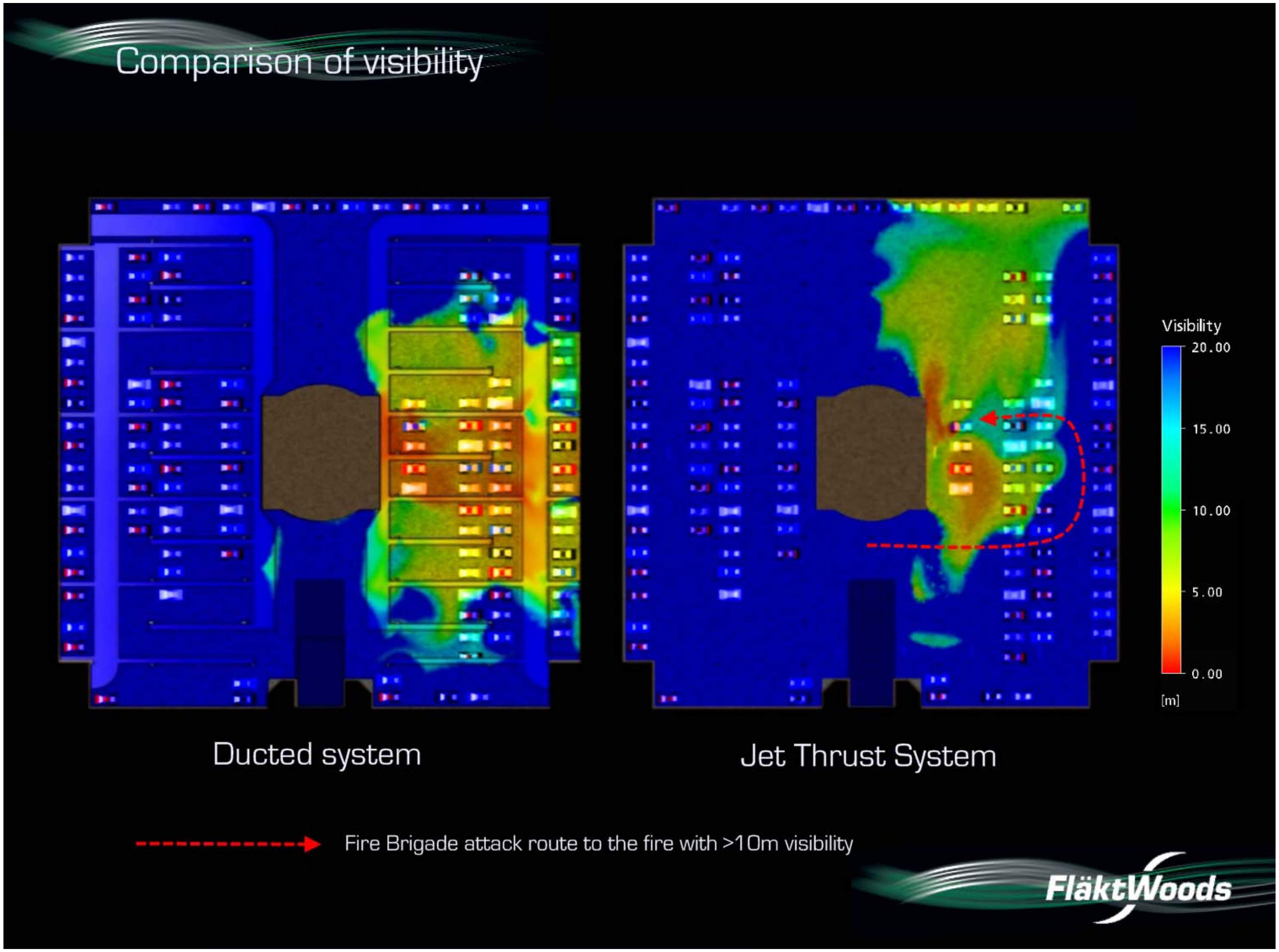


Applicable to Flakt Woods products only

Part 4

How to design for smoke control (specific to QCDD FSS-7.2)

Improving smoke control using Thrust fans



Qatar civil defence requirements (FSS 7.2)

- All projects in Qatar require a performance based design when designing with Thrust Fans. Ducted can be based on 10 ac/h using NFPA 88A and ASHRAE as reference.

ref. *Civil defence department minimum standards.*

- 4MW or 8MW design fire dependant on whether sprinklers
- 6 ac/h for pollution venting

Qatar civil defence requirements (FSS 7.2)

- Delivery vehicles….Design fire must increase to 10 MW or higher
- Design fire must be flaming polyurethane (Dense plastic).
- Design fire considered at zone boundaries (most onerous). Justification must be provided.

Qatar civil defence requirements (FSS 7.2)

- Duration of CFD simulation must be 30 mins.
- Grid size must be a maximum of 0.2m x 0.2m x 0.2m within 10 metres of the fire and a maximum of 0.4m x 0.4m x 0.4m for all other areas.
- Sensitivity study to be carried out to show loss of jet fan nearest fire does not impact the design

Qatar civil defence requirements (FSS 7.2)

- Exhaust fans must be configured so that loss of a single fan will not reduce airflow by more than 50%.
- Exhaust fans must have a backup power supply such as emergency generator.
- Activation of system must be automatic with manual override facility.

Smoke control – Requirements (Occupants)

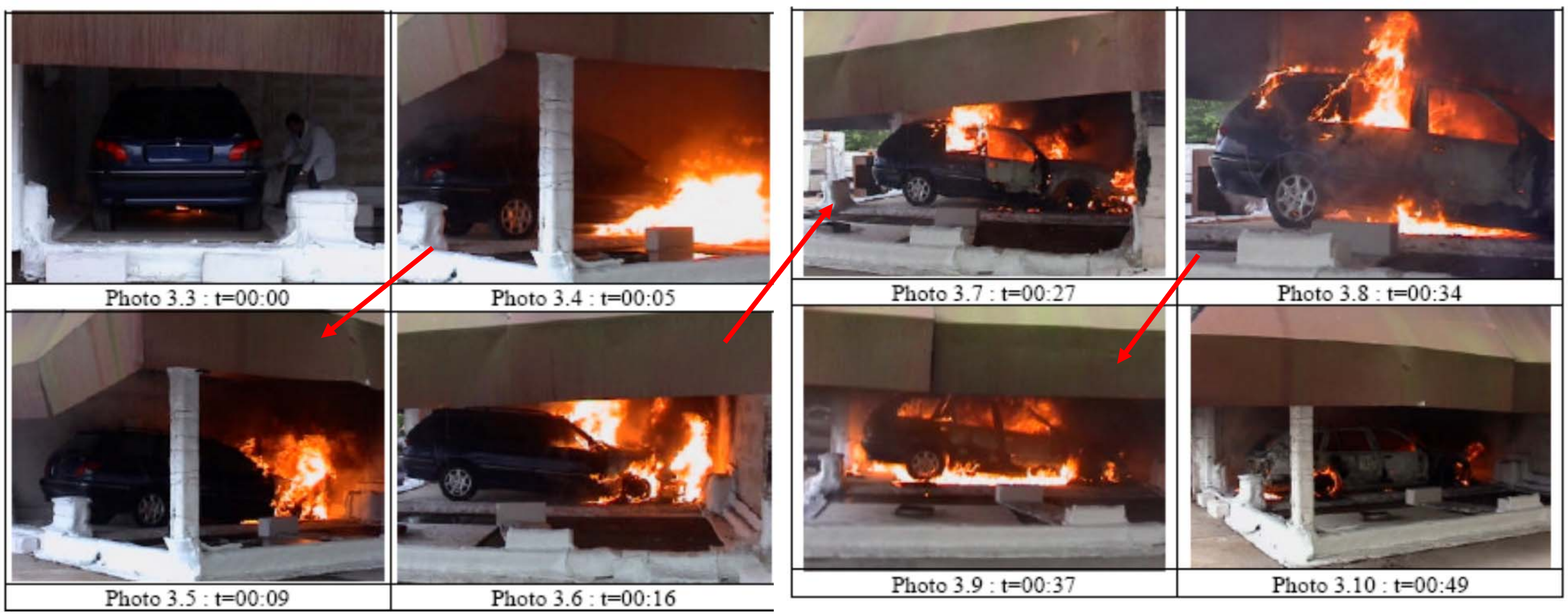
Element	Acceptance Criteria	Validation
Visibility	▪ 10 m minimum visibility	Internationally recognised criteria for areas where the occupants would be familiar with their surroundings.
Temperature	▪ 60º C air temperature to lower layer	Referenced in BS 7974 as a valid acceptance criteria.
Radiation	▪ 2.5 kW/m² maximum radiation from hot layer at a height of 2.5m.	Referenced in Fire Engineering Guidelines, Fire Code Reform Centre, Sydney, Australia, 1996.
		Supported by the research of Jin, T., Chapter 2-4, SFPE Handbook, 3rd edition, NFPA, Quincy, Mass, USA 2002.

Smoke control – Requirements (Fire brigade)

Element	Acceptance Criteria	Validation
Visibility	■ 10 m minimum visibility	Accepted tenability criteria for Fire fighters in protective clothing and breathing apparatus taken from the Australasian Fire Authorities Council Fire Brigade Intervention Model for a period of ten minutes.
Temperature	■ 120º C air temperature to lower layer	
Radiation	■ 3.0 kW/m² maximum at 2.5 m temperature above floor level.	

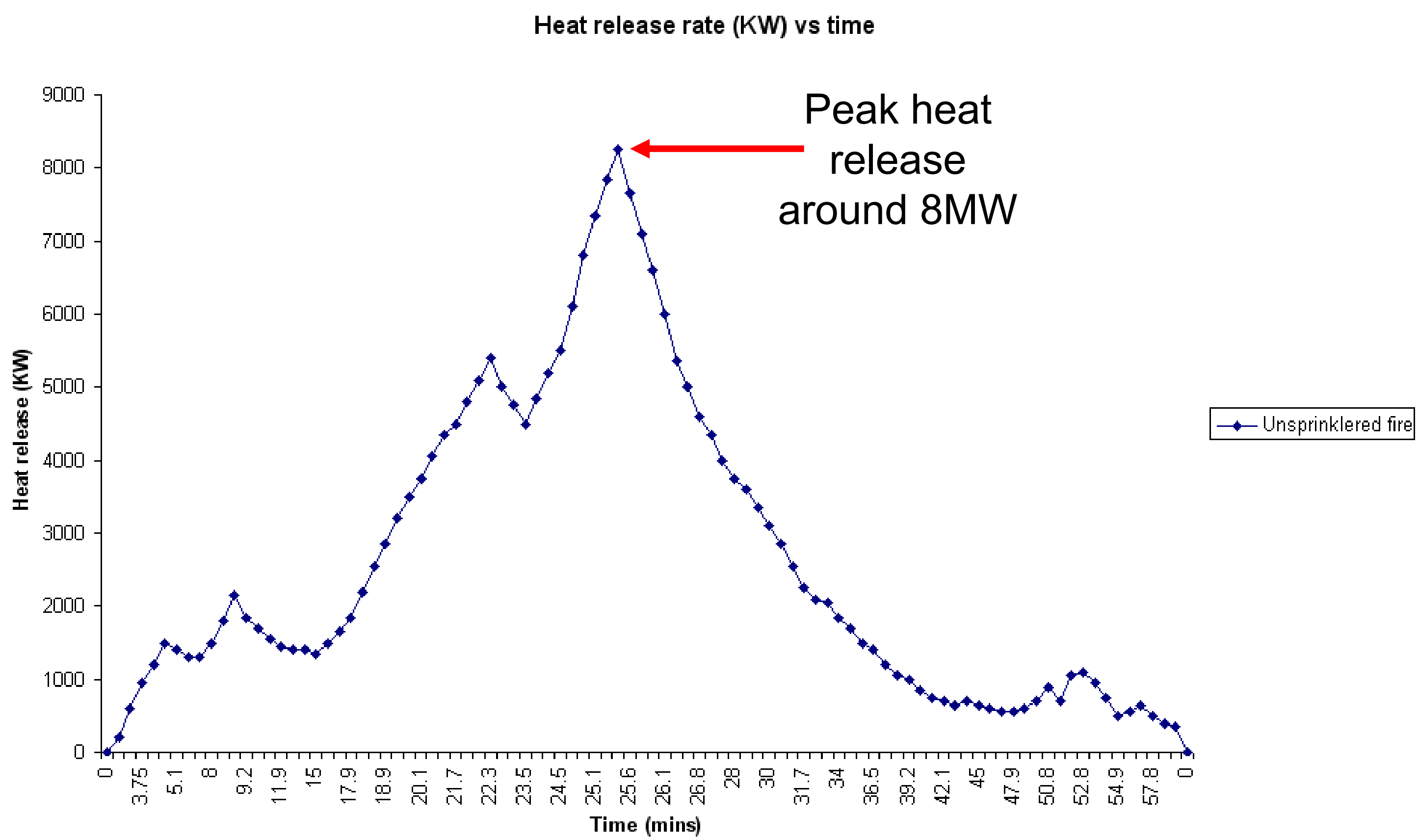
Fire test(s) completed by CTICM in 1995

- Controlled fire tests under calorimetric hood.
- 1995 Peugeot 406 estate car was used.
- **A mean heat of combustion of 26.3MJ/kg** was derived from the mass lost and energy released during first 35mins.



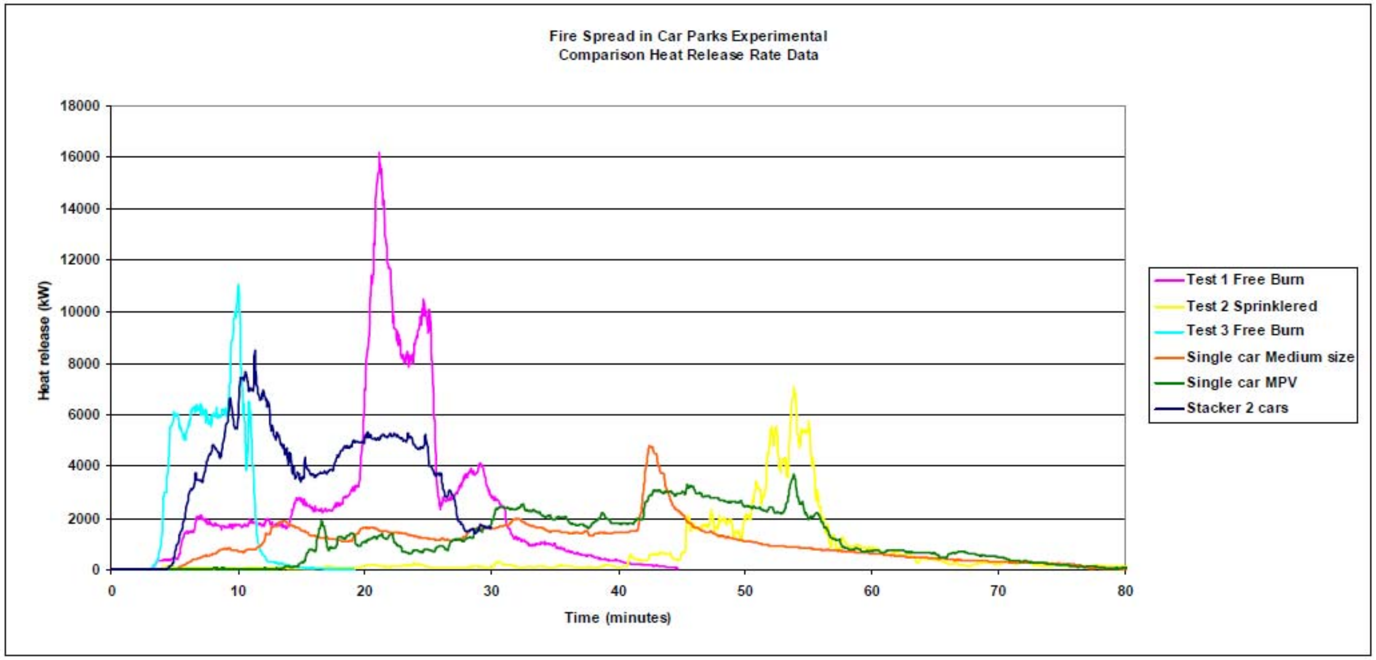
Ref. CTICM report 2001, **INC 01/410b DJ/NB**.

Maximum measured heat release rate



• Graph shows peak heat release rate from tests performed 1998 to 2001 (CTICM, France). **No sprinklers** involving 3 cars (Renault Laguna).

More recent research by BRE



- **16MW with no sprinklers!** (3 car fire) – test 1
- **7MW with sprinklers** but after 50-60 minutes into fire – test 2
- **5MW** single car (medium sized) after 40 minutes
- **4MW** single car (Multi purpose vehicle) after 40 minutes

Why are we seeing an increase in fire size?

Materials used in automobiles

Steel sheet	41 %
Plain steel	18 %
Aluminium	8 %
Cast iron	6.4 %
Zinc, copper, magnesium	2 %
Rubber	5.6 %
Plastics	**9.3 %**
Other materials*	9.7 %

**Adhesives/paints 3.0 %, glass 2.9 %, textiles 0.9 %, fluids 0.9 %, miscellaneous materials 2 %*

Plastics used in a typical car

Component	Main types of plastics	Weight in av. car (kg)
Bumpers	PP, ABS, PC/PBT	10.0
Seating	PUR, PP, PVC, ABS, PA	13.0
Dashboard	PP, ABS, SMA, PPE, PC	7.0
Fuel systems	HDPE, POM, PA, PP, PBT	6.0
Body (incl. panels)	PP, PPE, UP	6.0
Under-bonnet components	PA, PP, PBT	9.0
Interior trim	PP, ABS, PET, POM, PVC	20.0
Electrical components	PP, PE, PBT, PA, PVC	7.0
Exterior trim	ABS, PA, PBT, POM, ASA, PP	4.0
Lighting	PC, PBT, ABS, PMMA, UP	5.0
Upholstery	PVC, PUR, PP, PE	8.0
Liquid reservoirs	PP, PE, PA	1.0
Total		**105.0**

Source: http://www.plasticsconverters.eu

- Increased use of plastics to reduce weight and capital cost of production.
- Vehicles generally larger - more plastics required to reduce weight i.e. plastic car body panels.

Current Standards / Guidance

QCDD FSS7.2 and BS7346 Part 7 recommends:

• 4 MW for single car (sprinklers)

• 8 MW for two cars (no sprinklers) however latest research shows it can be much higher

Table 1 **Steady-state design fires**

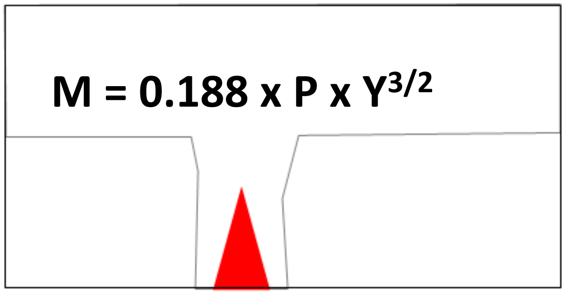
Fire parameters	**Indoor car park without sprinkler system**	**Indoor car park with sprinkler system**
Dimensions	5 m × 5 m	2 m × 5 m
Perimeter	20 m	14 m
Heat release rate	8 MW	4 MW

Time-dependent design fires should be based on an experimental test fire, which should be described and justified in the documentation specified in Clause **18**. Where the experimental data has been placed in the public domain a reference to the publication may be used as justification.

Neither QCDD FSS7.2 or BS7346 Part 7 informs the reader of how to design a smoke control system. Only performance objectives and guidance are given.

Tools available to the designer

Three categories: EMPERICAL, ZONE MODELS, CFD

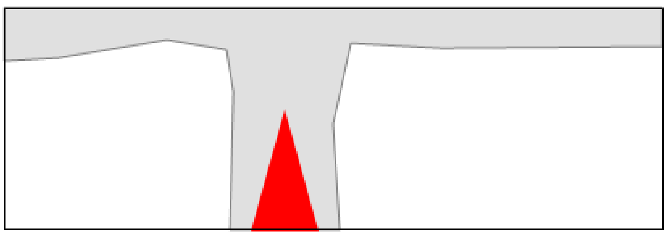


Thomas, Alpert et al

HAND CALCULATION (EMPERICAL):

Very simple

Applicable to **limited range of conditions**



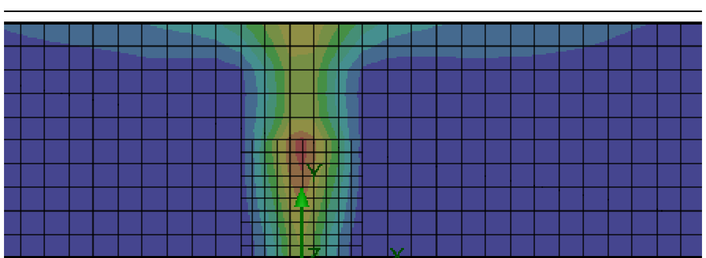
CFAST. Ozone

ZONE MODELS:

Simple and fast to use

Application limits defined

Multi compartments



ANSYS, FLUENT, FDS, Flosim

CFD MODELS:

Complex

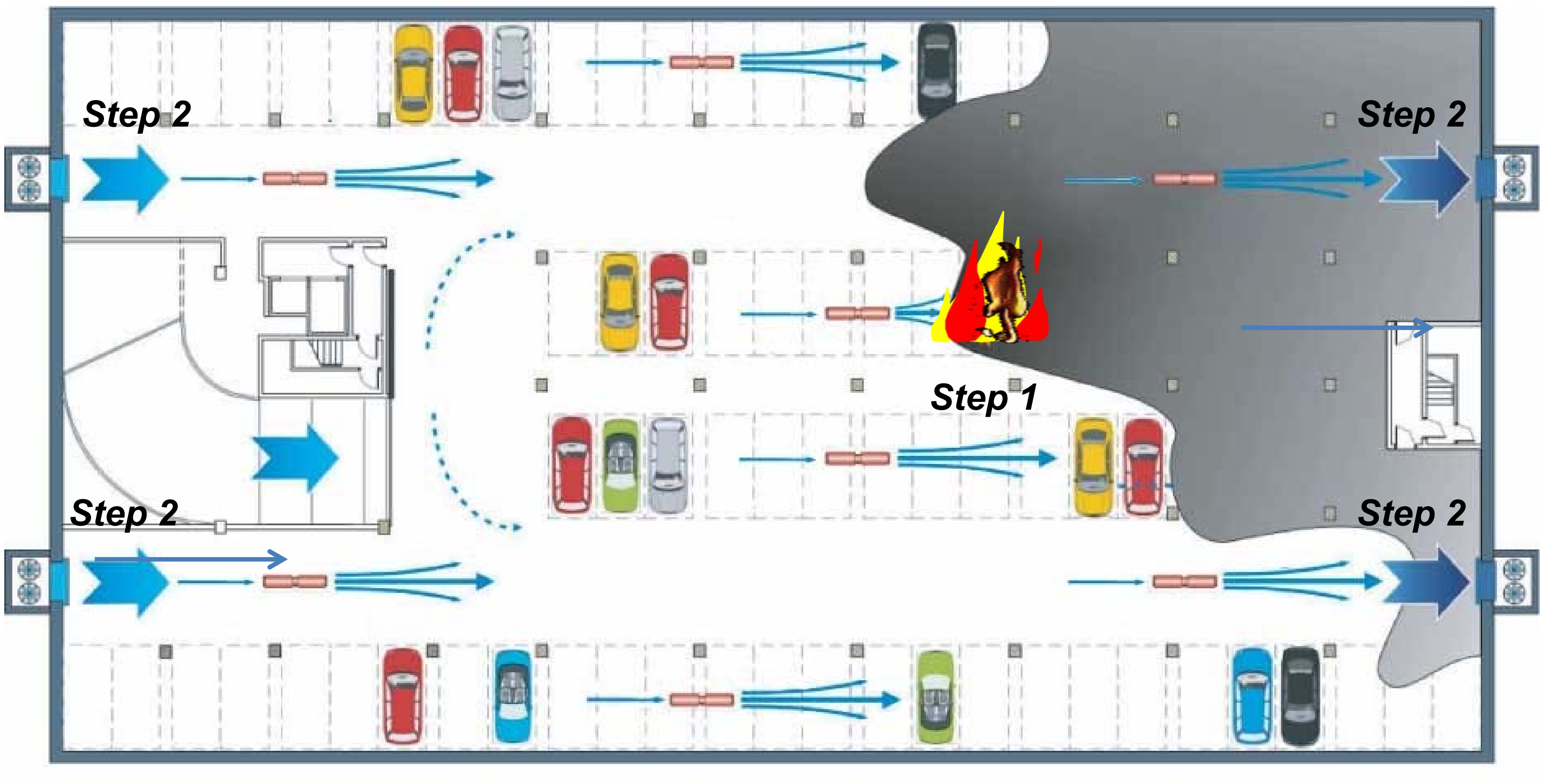
Expert knowledge

Most accurate but costly

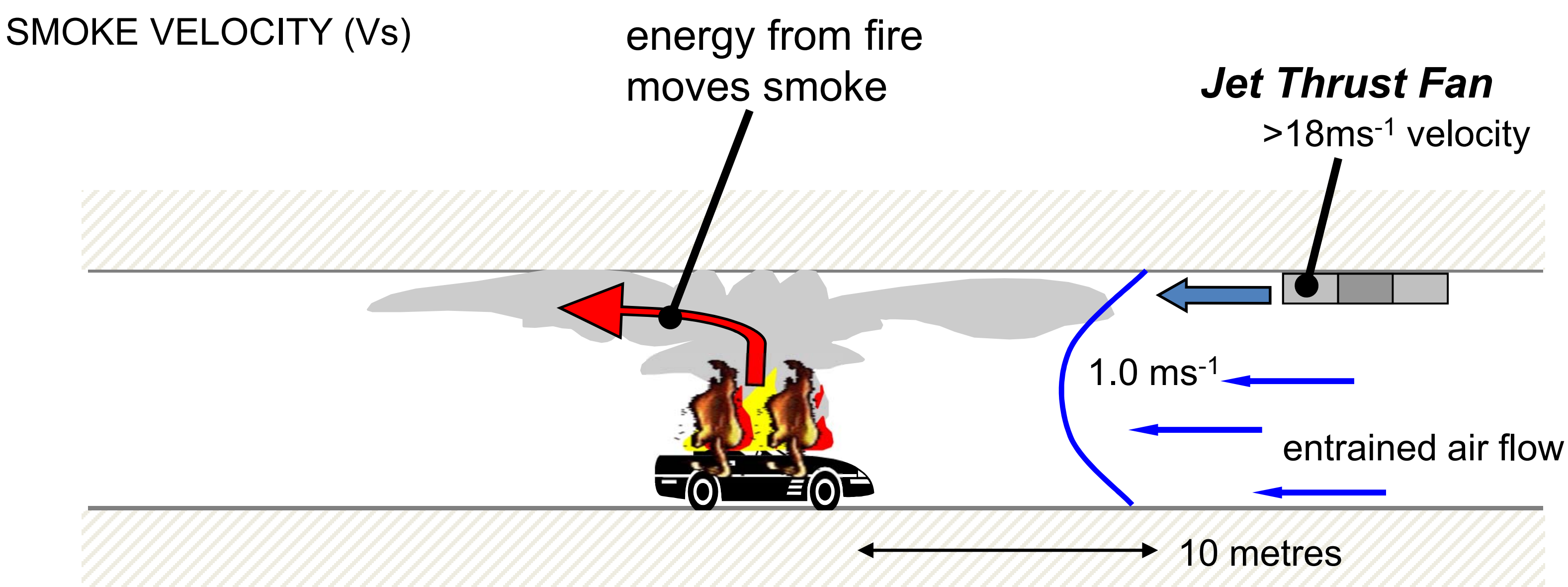
How to design for smoke control - Design steps

1. Determine design fire size according to whether or not there are sprinklers –SLIDE
2. Determine zone layout, at least one extract and one supply point per zone. Decide on general flow distribution and smoke travel distance
3. Calculate the velocity of smoke at required control distance (10 metres upstream)
4. Calculate the minimum design flow rate for smoke control
5. Calculate the mass-flow of smoke and smoke temperature at the fire
6. Calculate mass-flow towards the extract
7. Calculate density downstream
8. Calculate the extract flow rate required
9. Calculate Thrust fan quantity
10. CFD analysis

Steps 1 and 2: Fire size and ventilation configuration



Step 3: Smoke control calculations



Step 3: $V_{ceiling\ jet} = 0.195 * Q^{1/3} * h^{1/2} / r^{5/6}$ (Albert et al)

1 car = 0.6m/s at 10 metres from the fire (3 metres high car park)

2 cars = 1 m/s at 10 metres from the fire (3 metres high car park)

Step 4: Minimum design flow rate

Design flow rate (Qd) = (Design width x height) x Min velocity

Min design velocity = $V_{ceiling\ jet}$ / (installation effect x velocity effect)

Where:

Velocity effect = 1 – $V_{ceiling\ jet}$ / V_{jetfan}

Step 3

Design width varies according to scenario but suggested starting point is 28 metres

(approx. 2 roadway widths + 2 car park spaces length).

Typical installation factors:

- 0.9 with pillars and no down-stand beams
- 0.7 with pillars and down-stand beams where the spacing of beams > 18 fan diameters
- 0.5 with pillars and down-stand beams where the spacing of beams <= 18 fan diameters

Smoke calculations

The remaining calculation methods (steps 5 to 9) are presented in a technical paper. Please contact your Fläkt Woods representative for more information.

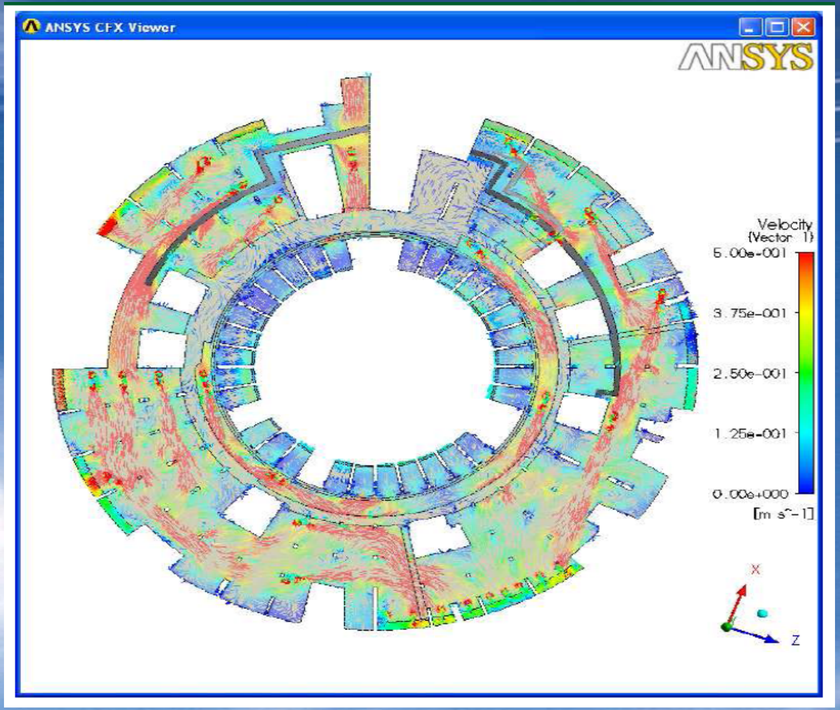
Closing statement

The method presented, developed by Mr J. Allen, has been **proven to work on a number of projects** however it is still relatively new. It is not presented within any standards however a technical paper has been written. The technical paper will soon be available through the Fläkt Woods website. Further research and development is needed to fine tune the approach.

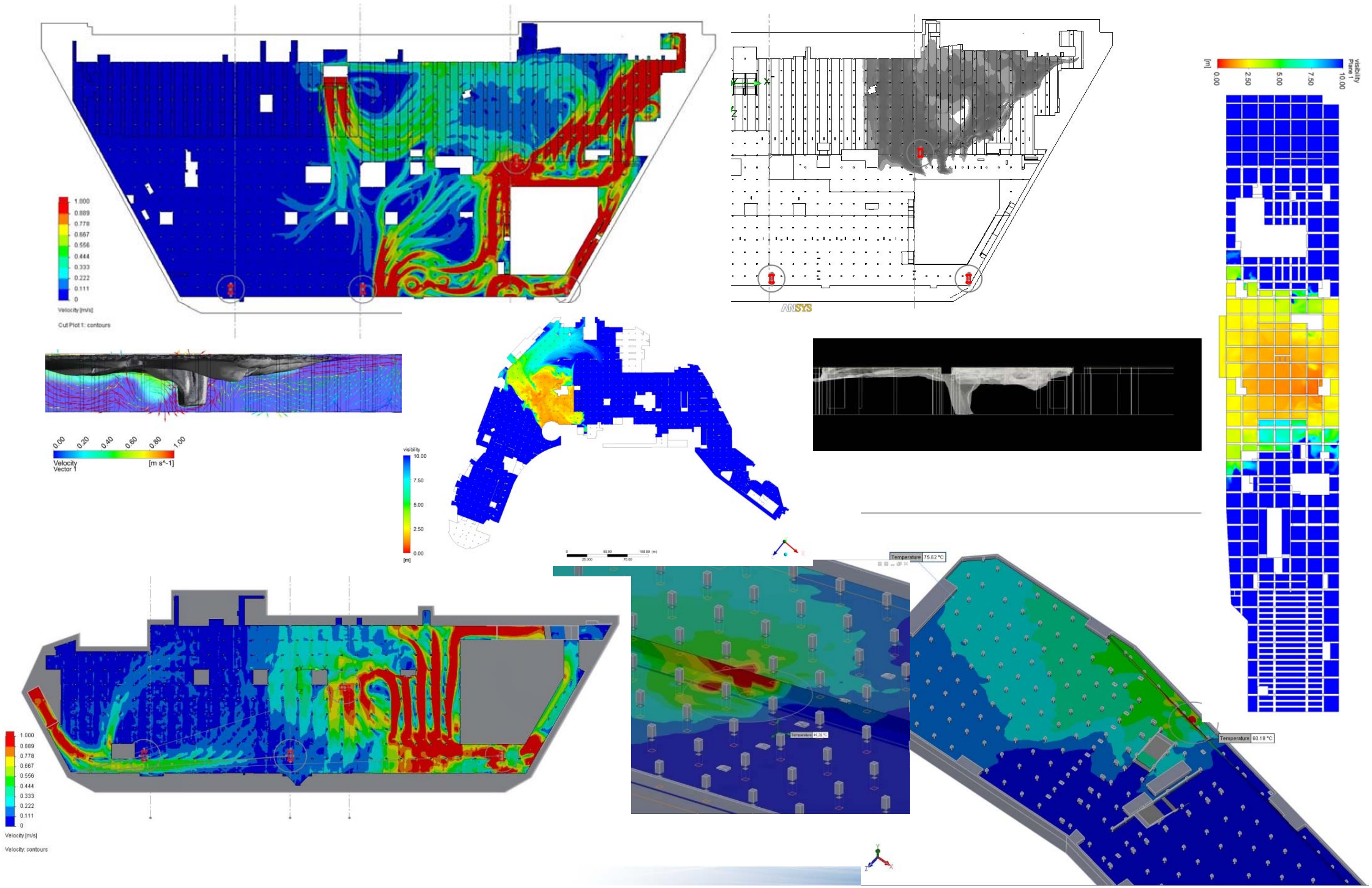
As stated by Dr H. P. Morgan, co-author of ‘Extending the principles of Impulse ventilation in tunnels to apply to smoke control in car parks’:

“ If the induced bulk air volume flow is greater than the extract volume flow rate, the discrepancy between what is being “pushed” towards the exhaust and what is being removed must somehow travel back past the fans to become available at the fan inlets. This can either take the form of a recirculation pattern <u>throughout the car park</u>, causing smoke to affect the areas intended to be kept clear, or it can take the form of a local recirculation at each fan, which would have less of an adverse effect on system performance. The significance of these recirculation patterns cannot be assessed by zone- model methods but should be revealed by CFD modelling”

Part 5
Using Computational Fluid Dynamics (CFD)

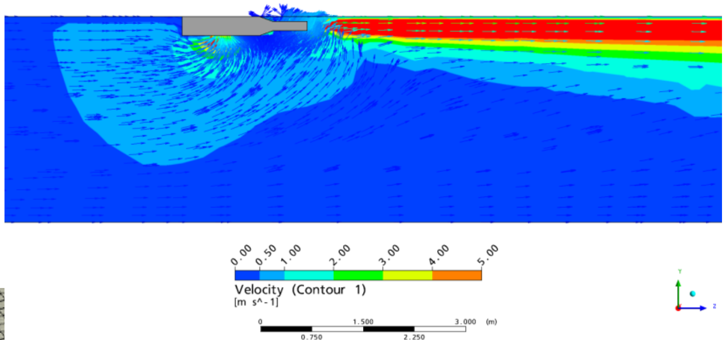


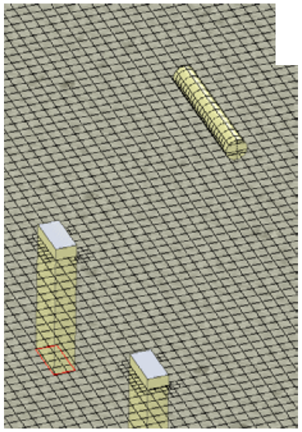
Use of CFD – Air distribution & smoke control

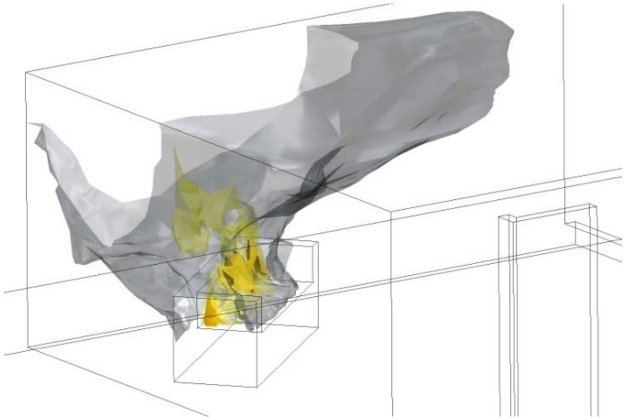


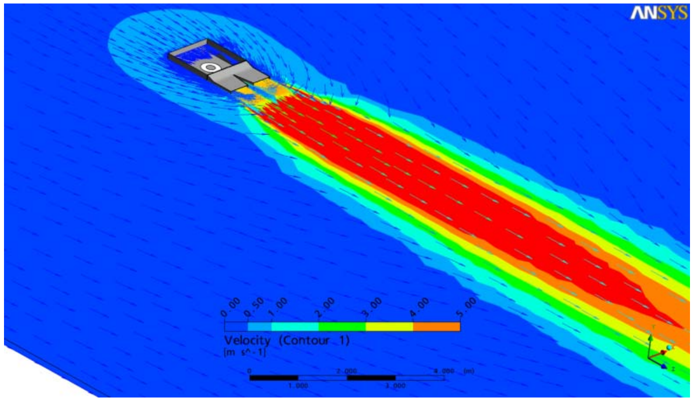
CFD Software types

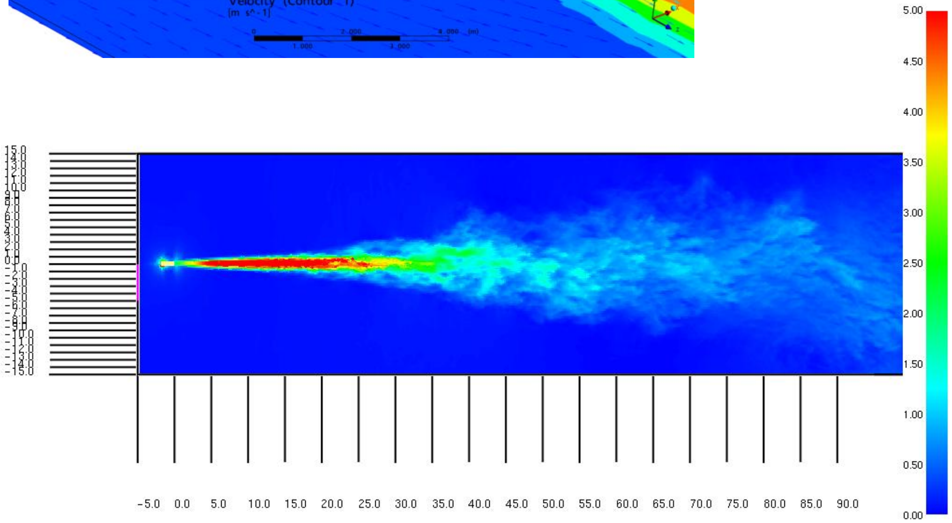
CFX, Fluent, Flosim, Phoenics - RANS
FDS, CFX - LES
CFX - DES

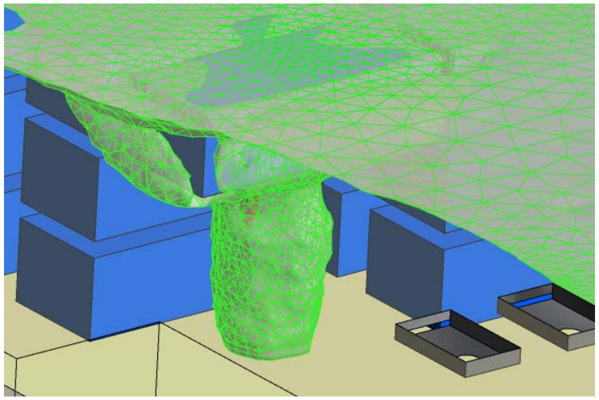












FläktWoods

Is all CFD software fit for purpose?

ITF 75 – Model in FDS5

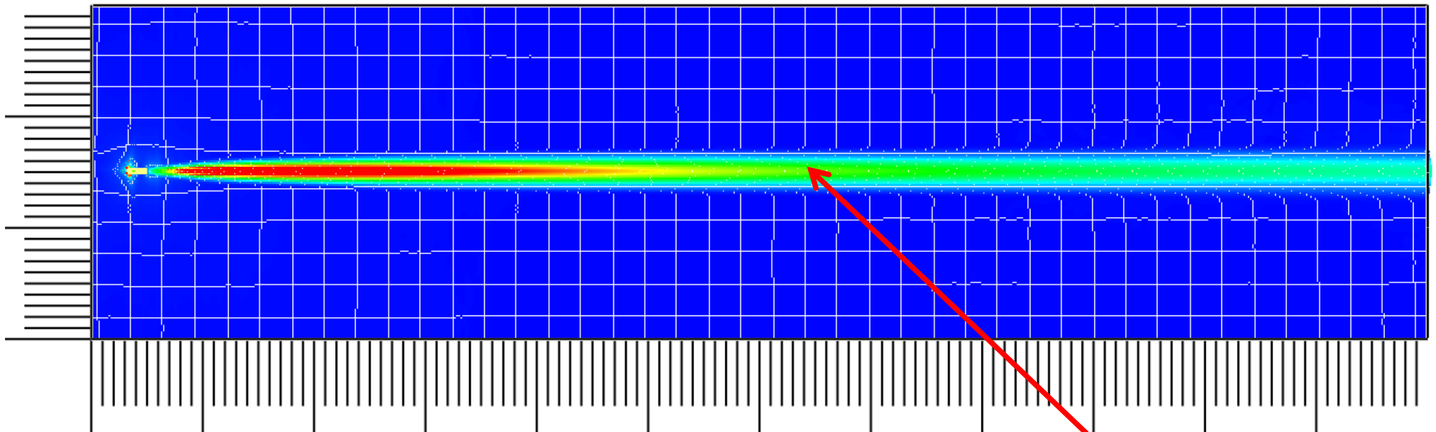
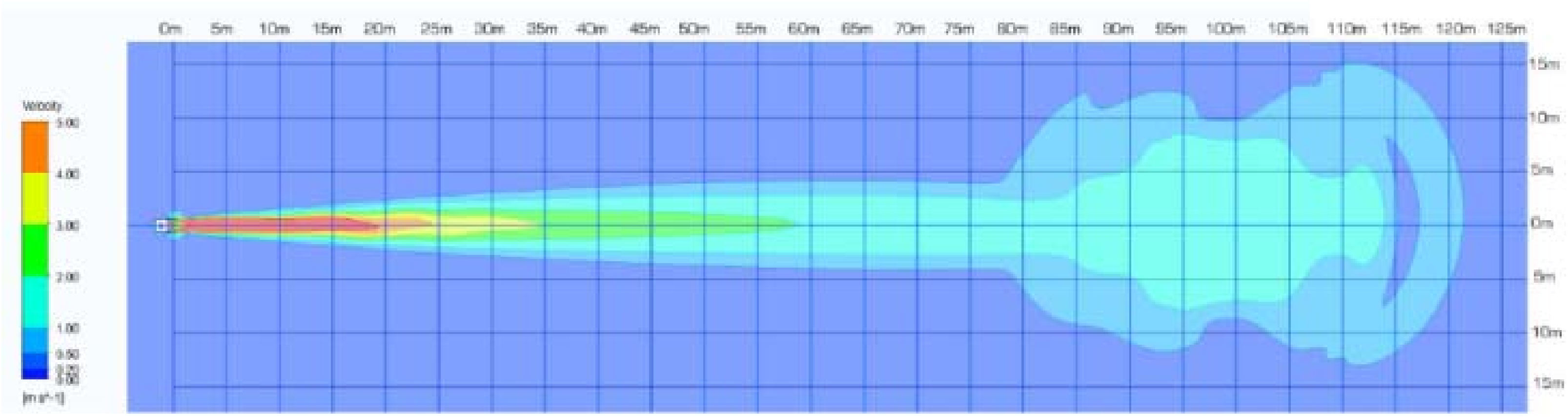


Image courtesy of independent fire consultant

20cm mesh!

ITF 75 – Model in CFX (SST turbulence model)

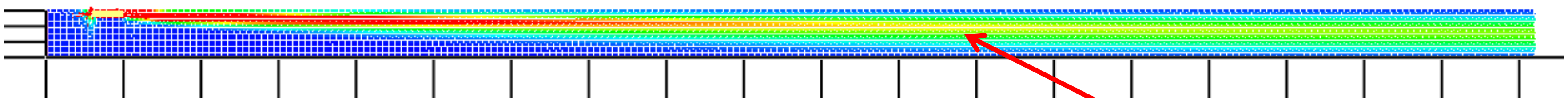


Is all CFD software fit for purpose?

ITF 75 – Model in FDS5

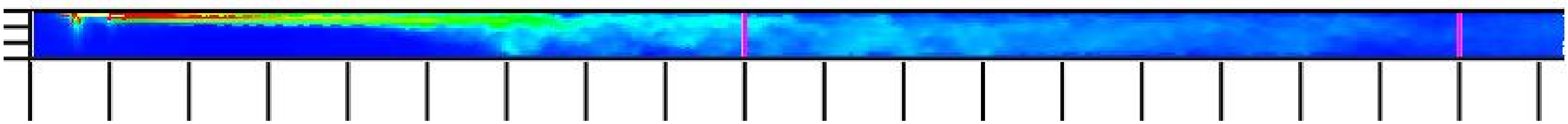
Images courtesy of independent fire consultant

FDS5, horizontal

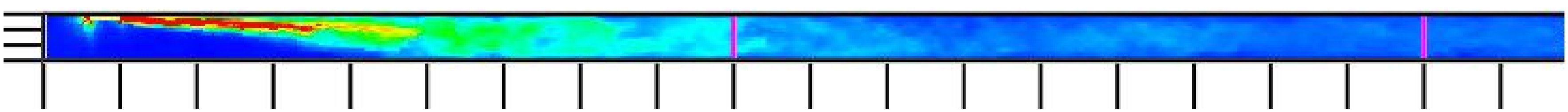


20cm mesh!

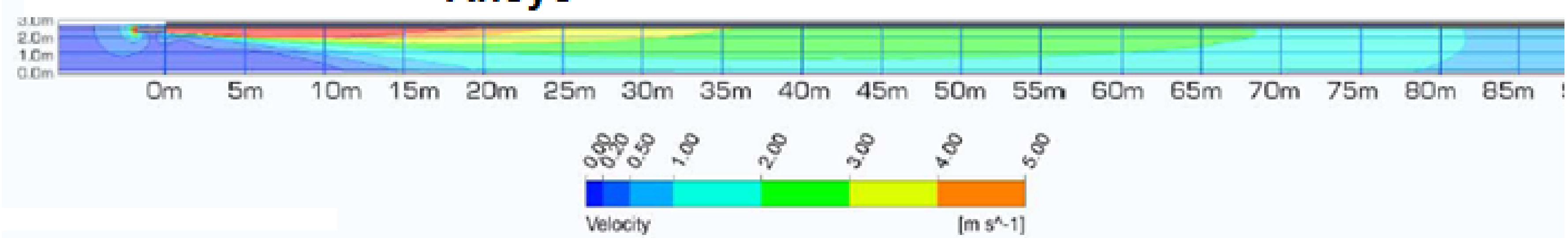
FDS6 - horizontal



FDS6 - 5 degrees down



Ansys



Is all CFD software fit for purpose?

Images courtesy of ANSYS

Figure 1: Streamlines for plane diffuser flow for both models. The SST model predicts the separation zone in close agreement with data, whereas the k-ε model fails to capture the physics of this flow entirely.

- **CFX (RANS based) using SST turbulence model predicts separation and reattachment of flow where as the standard K-e model fails to capture this flow entirely**

Is all software fit for purpose?

Conclusion:

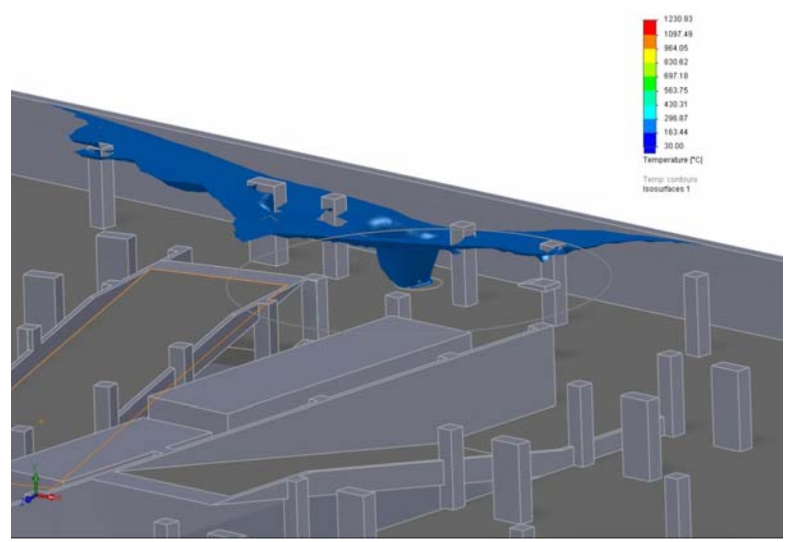
• **FDS5 severely over predicts** the performance of jet fans. This is due to turbulence model used (no separation and reattachment)

• **FDS6 is much better** (due to new turbulence model) but under-predicts flow. Conservative estimate?

• **CFX (RANS based model) provides the closest match to test data** providing appropriate mesh sizes are used AND suitable turbulence model (SST is best at simulating separation and reattachment of flow)

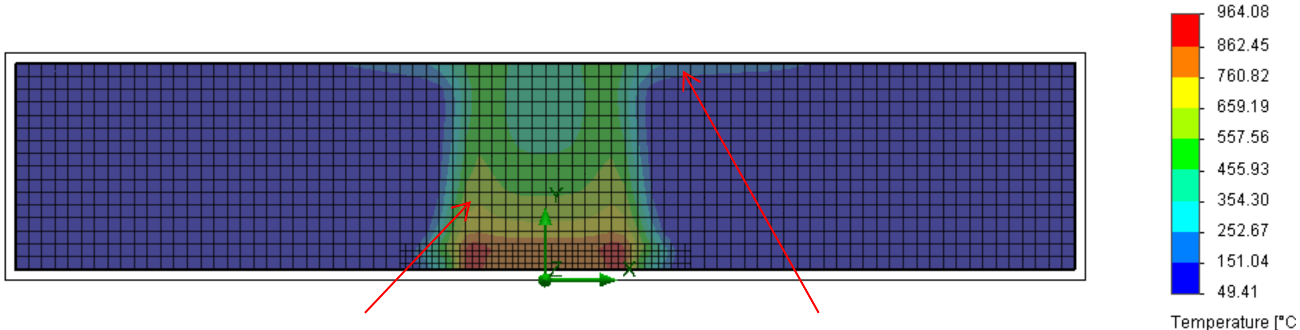
‘Inert’ volumetric heat source model

- Volume of heat release must be specified by user
- Verification of volume essential prior to running simulations
- Uniform distribution of heat and smoke
- Simple / less computationally expensive
- Ideal for large complex spaces i.e. car parks
- Only suitable for **well ventilated fires**
- Can be used with any radiation sub model
- Realistic temperatures and flow phenomenon are achievable



Using the correct fuel area / volume in CFD models

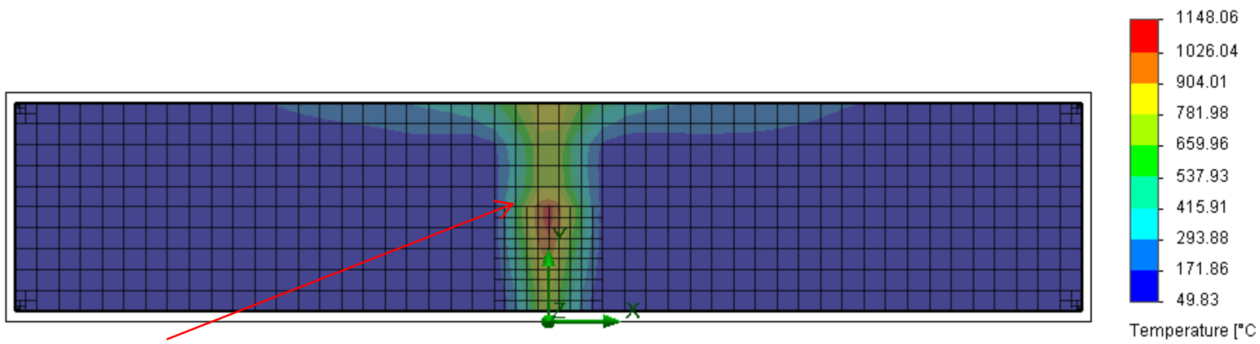
- **Simplest method is volumetric heat source**
- **Volume of source needs careful consideration**



Incorrectly defined heat source volume

Flame height too low **Layer depth too shallow**

More realistic heat source volume



Flame height & layer depth more consistent with empirical correlations

Volumetric heat source recommendations

- Fire power density in the range from 500 kW/m³ to 1000 kW/m³
Depending on application and geometry.

- Experimentation is required to reach realistic results.

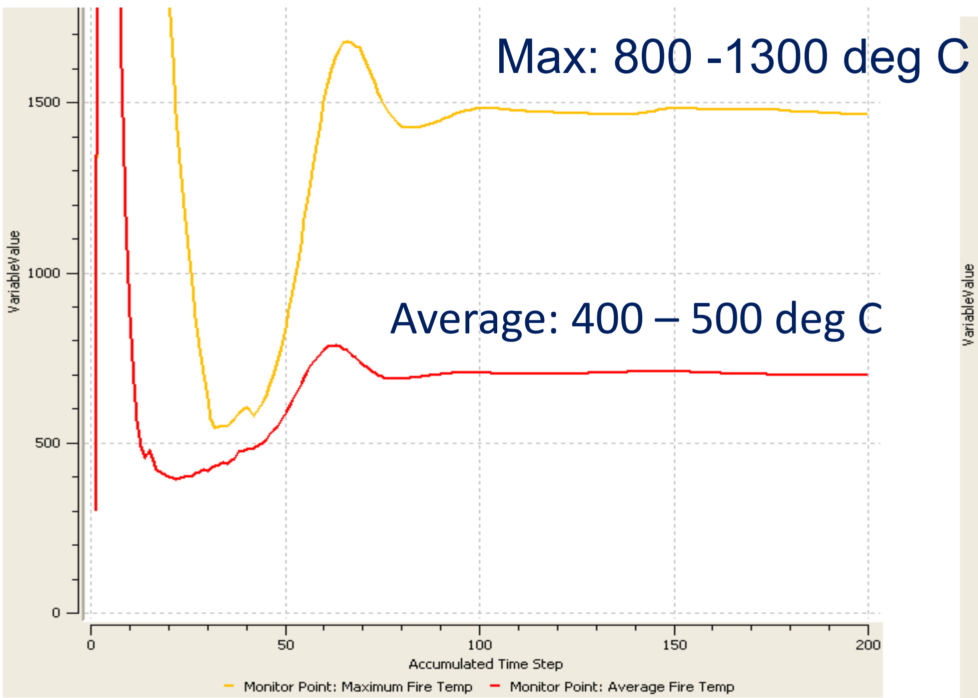
- Comparison should be made against calculations and / or
experiment in terms of both flame temperature and flame height

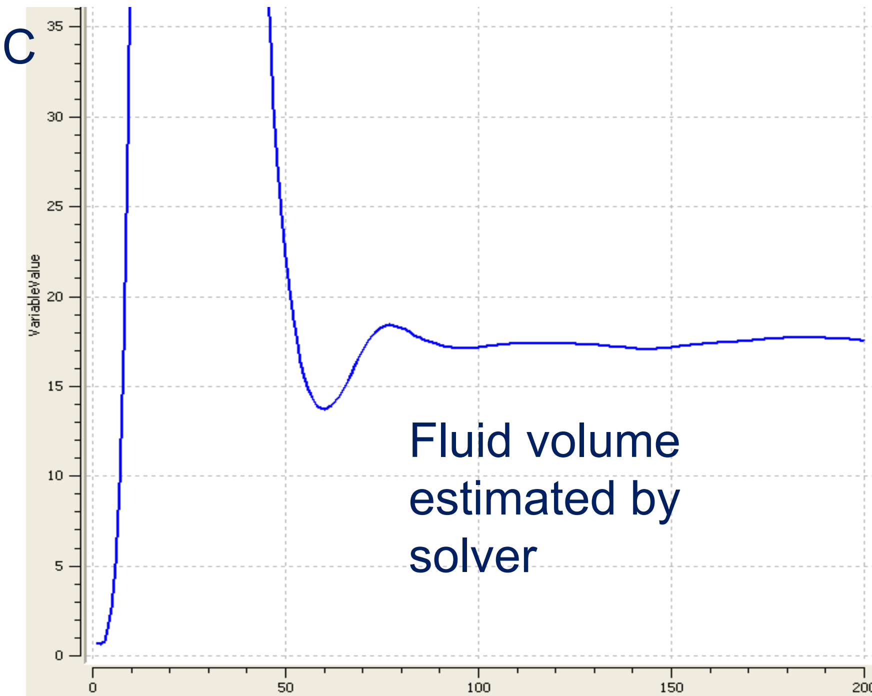
REALISTIC RESULTS OBTAINED SEVERAL METRES FROM THE SOURCE.

See ‘Treatment of fire source in CFD’ , R. Yan, V. Cheng, R.Yin, 2003

Inert models - Volumetric heat source

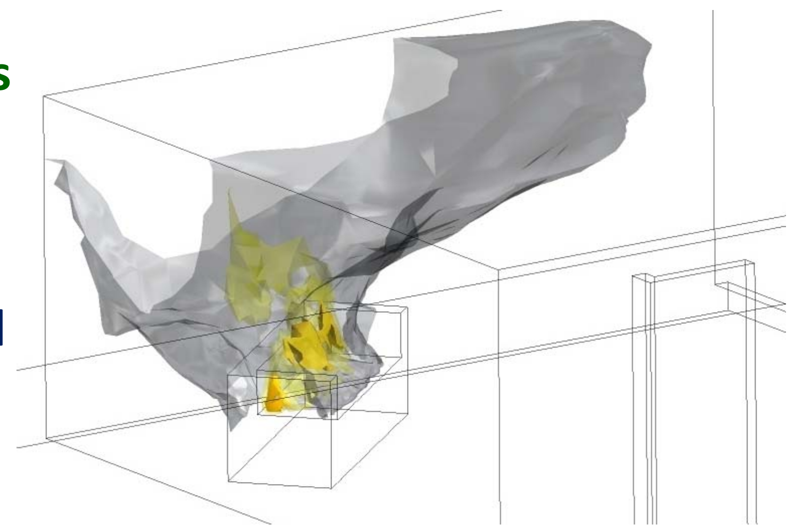
- Vitally important to correctly define the fire volume
- Good reference is 'Treatment of fire source in CFD' , R. Yan, V. Cheng, R.Yin, 2003



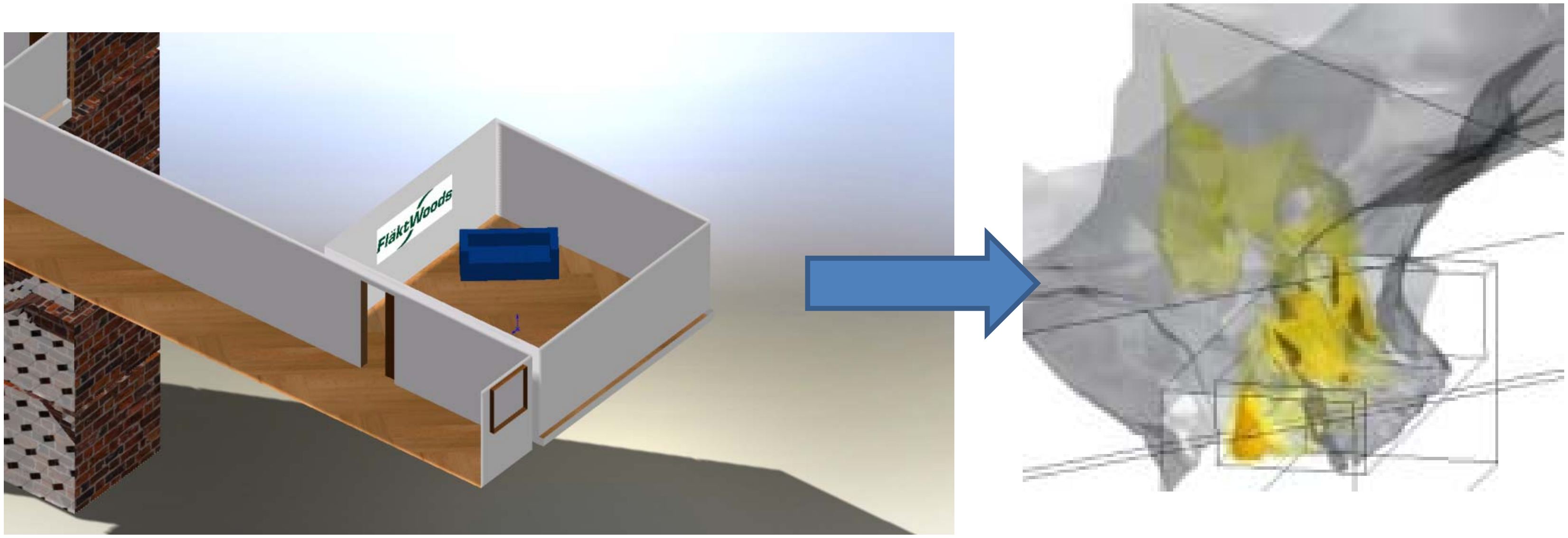


Reacting combustion model

- Aims to predict heat distribution in the flaming region
- Area of fuel source must be specified by the user
- Suited to simpler geometries as **computationally more expensive**
- **Significantly increases timescale from days to weeks**
- **The only choice for under-ventilated fires**
- Heat and smoke distribution non-uniform
- Can be used with any radiation sub model



Example of a reacting combustion model



- Defined as an ‘area’ source of combustible material

- CFX calculates the mixing of gaseous fuel with air and assumes the combustion rate is infinitely fast with respect to turbulent mixing

- The distribution of combustion in the fire plume is predicted

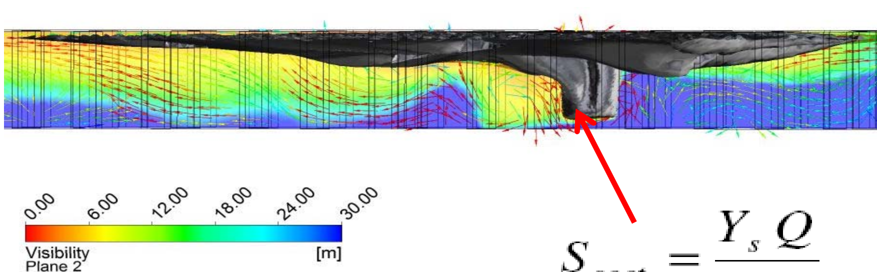
Estimation of smoke production

• Quantity of smoke and toxins prescribed by the user as a function of the mass of combustible products.

• Refer to smoke yield factors for common materials in fire engineering literature. Source: C.P. Bankston, B.T. Zinn, R.F.Browner, and E.A. Powel, Comb. and Flame, 41, 273 (1981).

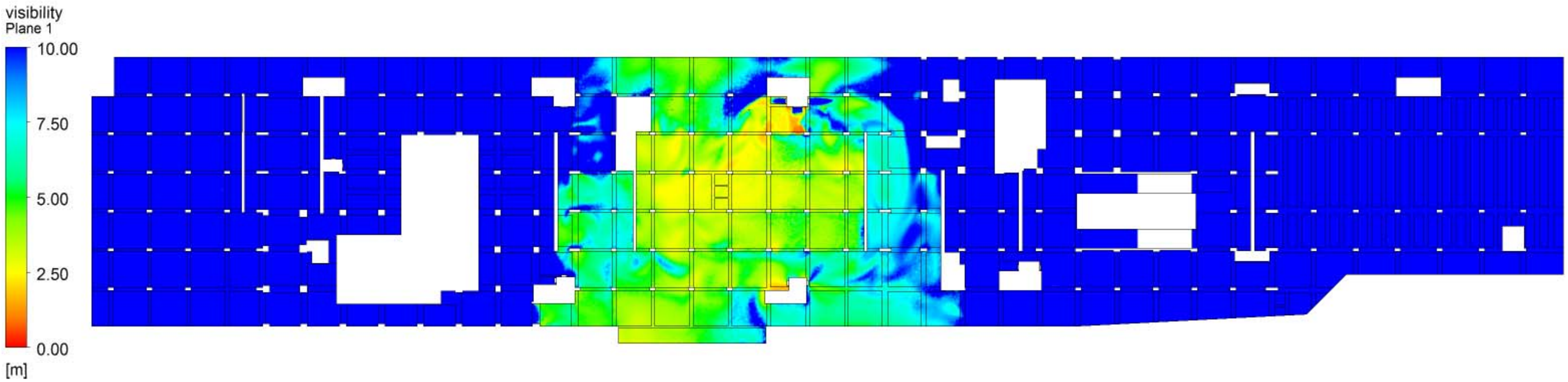
• Polyurethane fuel > 10 to 11% of the mass of combustible products. This falls mid way in the range which is typically 0.01 to 0.20 for flaming combustion.

• Characteristic heat of combustion (H_c) – typically 24 to 26 MJ/Kg.



$$S_{soot} = \frac{Y_s Q}{H_c}$$

Visibility through smoke



Light reflecting or light emitting correlations developed through scientific study.

S [m] = K / (α [m²/kg] x mass fraction [kg/kg] x density [kg/m³])

Ref. Klote J.,J. Mike, Principles of Smoke management, 2002

Where:

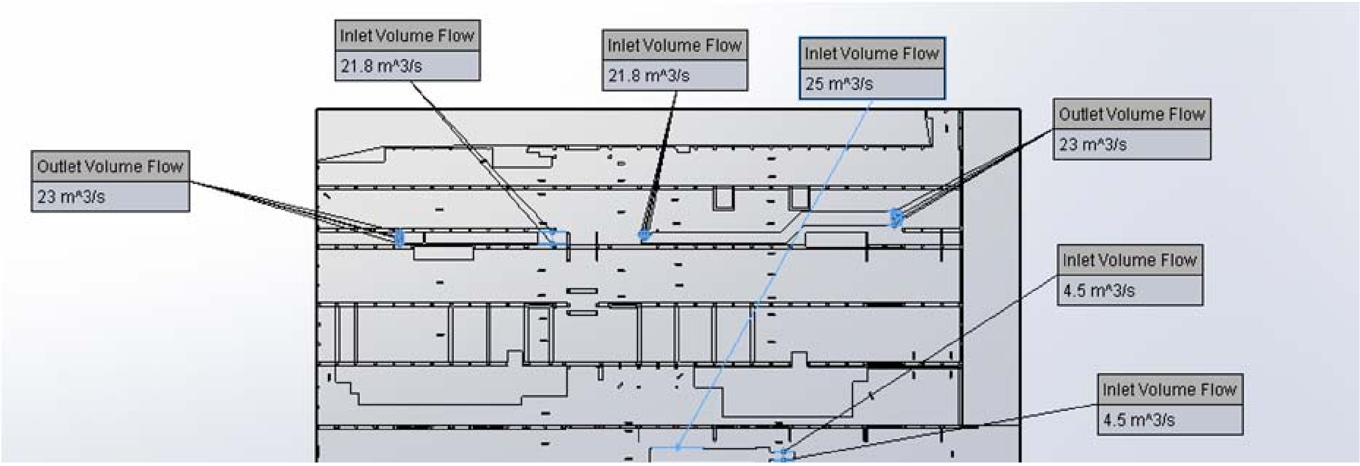
K = proportionality constant

Illuminating signs K = 8

Reflecting signs and building components K = 3

α = specific extinction coefficient (8700 [m²/kg] flaming combustion

Boundary and initial flow specification



Typical examples include:

• Initial flows present prior to the simulation (i.e. wind pressures)

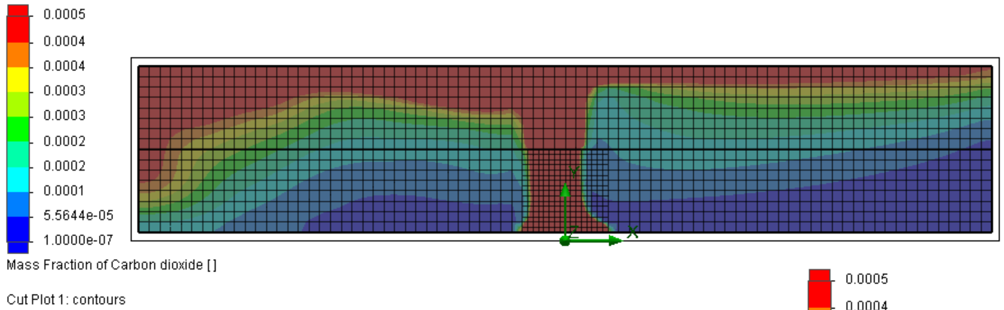
• Flows in / out through doors, windows, openings, vents or mechanical inlet / extract systems

• Change of momentum and / or energy in simplified representations of mechanical systems such as jet fans.

• Energy transfer (in the form of heat) at (to / from) walls.

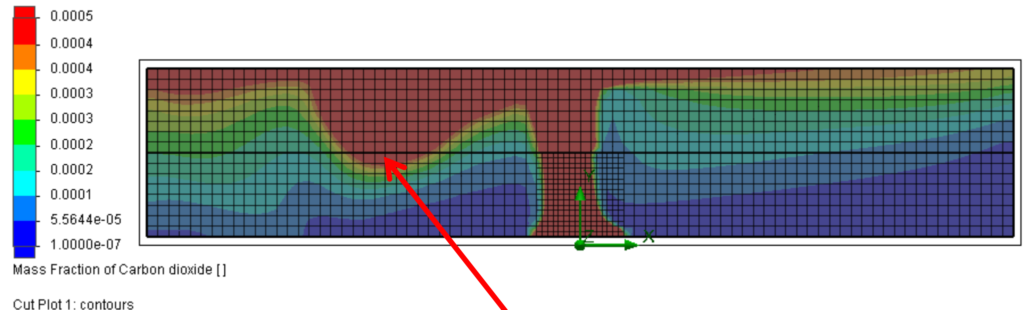
• Sources of mass, momentum and / or energy, e.g. at the fire, or through the release of a suppressant.

Importance of wall boundary specification



Adiabatic walls

Heat flux of 25W/m²/K + fixed temp



Smoke layer is lower!

The CFD user specifies how heat transfer is to be modelled at the walls:

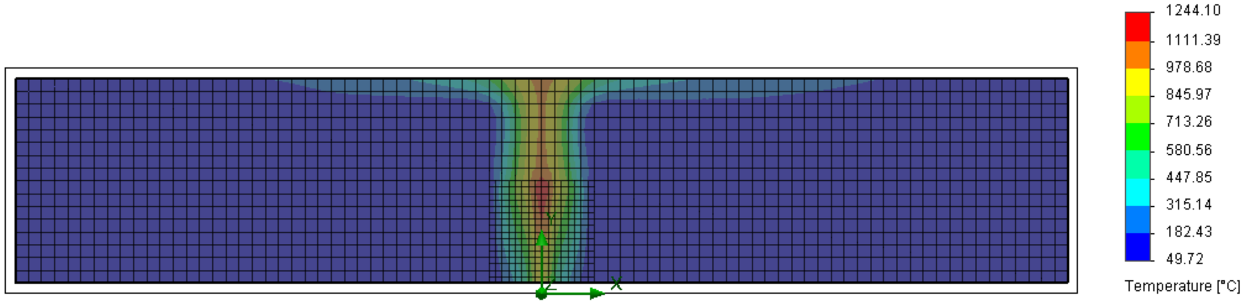
- Assume nil heat transfer, i.e. an adiabatic wall.

- Assume a constant wall temperature, leading to maximum rates of heat transfer. Suggestion use a heat flux - typically 25W/m²/K with a fixed temperature on the other side of the wall (Ref. EN191-1-2)

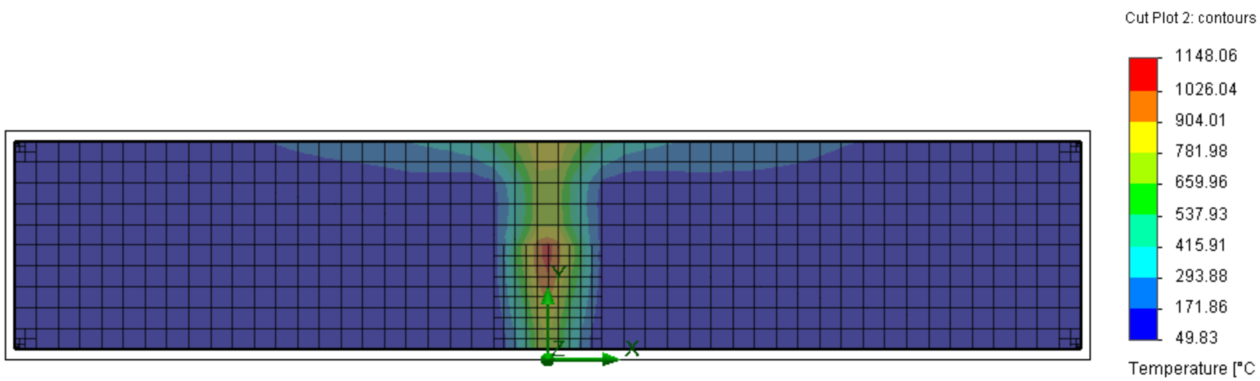
- Adiabatic wall condition **should be used with caution** since less smoke is predicted at lower levels and to faster smoke propagation towards extract.

Importance of the mesh size

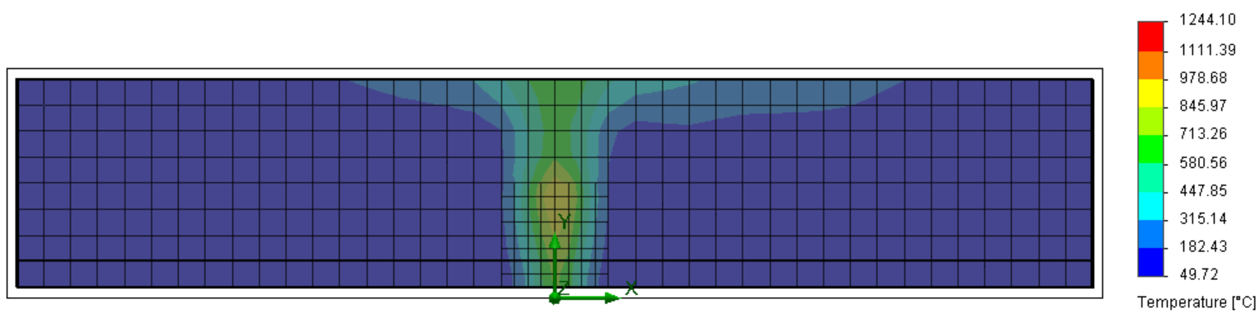
• Mesh resolution can have a significant affect on accuracy of predictions **BOTH** for RANS and LES models. LES models can be particularly sensitive.



0.25m
0.1m (fire region)



0.40m
0.2m (fire region)



0.50m
0.225m
(fire region)

Importance of the mesh size

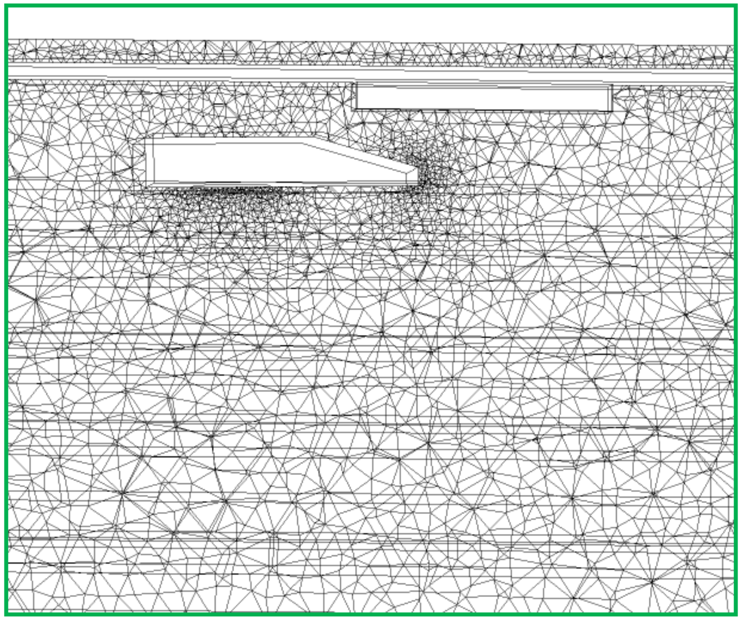
- Trade off between accuracy and run time.

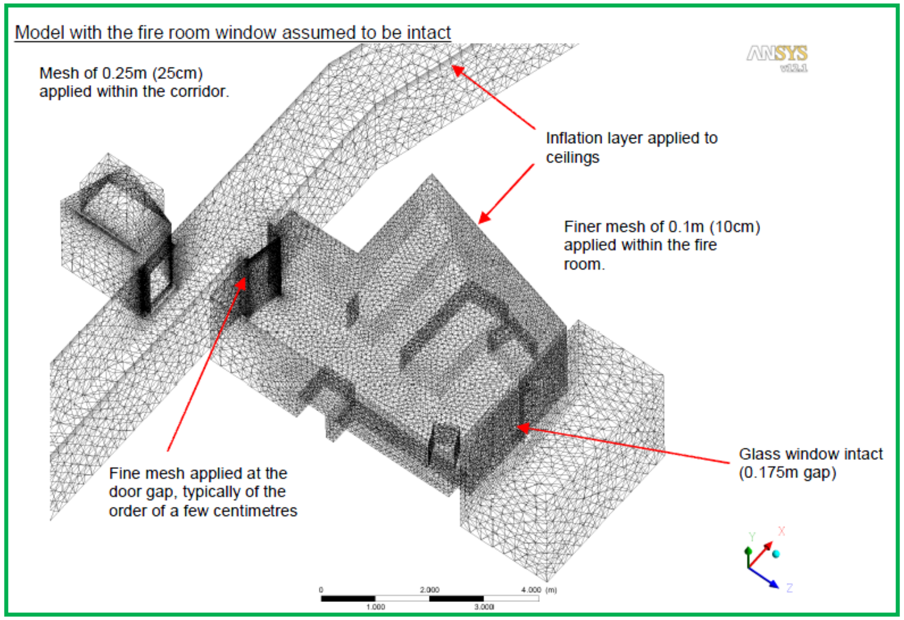
- Broad range of mesh sizes can be used – the user should ideally carryout a grid sensitivity check.

- Important flow regions such as fans and inlets / outlets should ideally remain constant using best practice guidelines.

- Inflation layers should ideally be used at wall interfaces although it is recognised this is not available in all software.

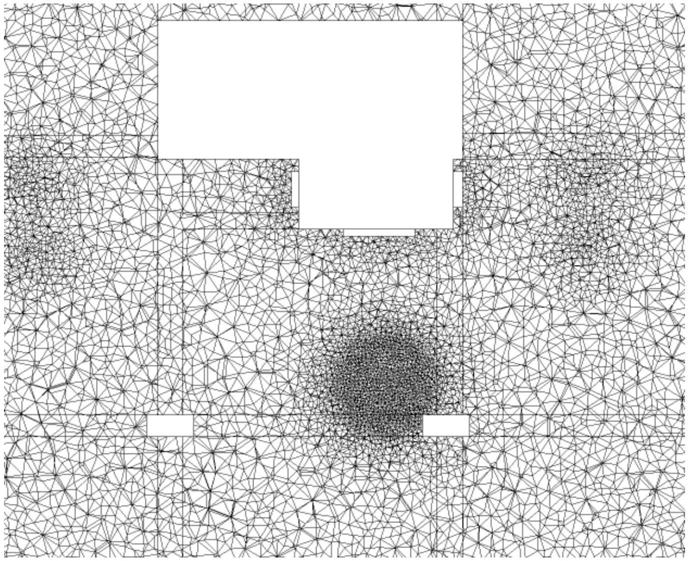
- Finer meshes should be used in the fire region to capture the complex heat exchange and flow phenomenon. Typically 0.1 to 0.2m will normally suffice.

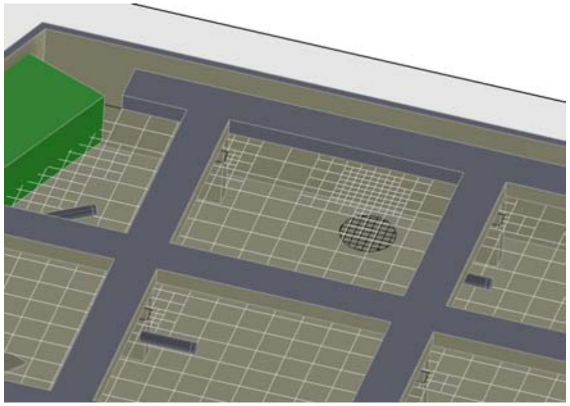


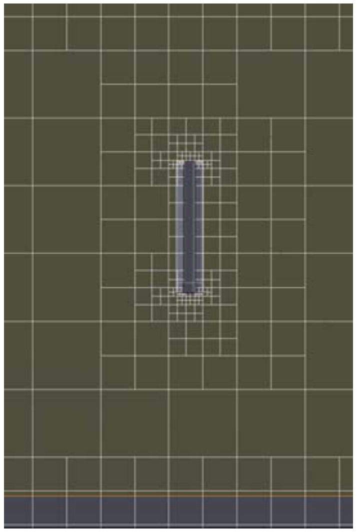


Importance of mesh aspect ratio

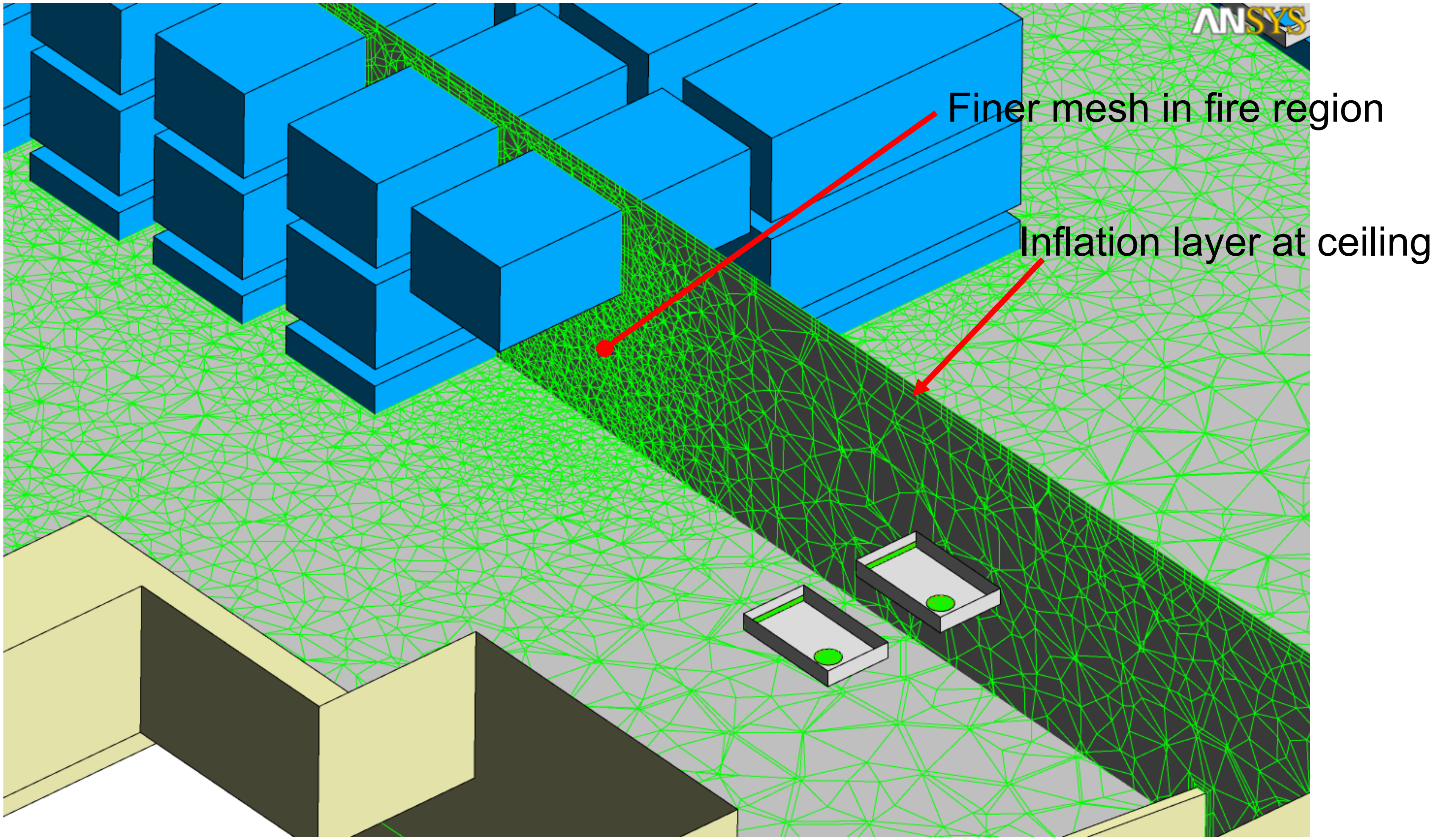
- Mesh aspect ratio can have a significant affect on accuracy of predictions **BOTH** for RANS and LES models. However LES can be more sensitive to large variations
  - COX and Kumar recommend 1 to 50 as max aspect ratio (RANS)
  - Lower aspect ratios close to the fire (1:1)
  - FDS models require much lower ratios, typically 1:3 (>timescale)



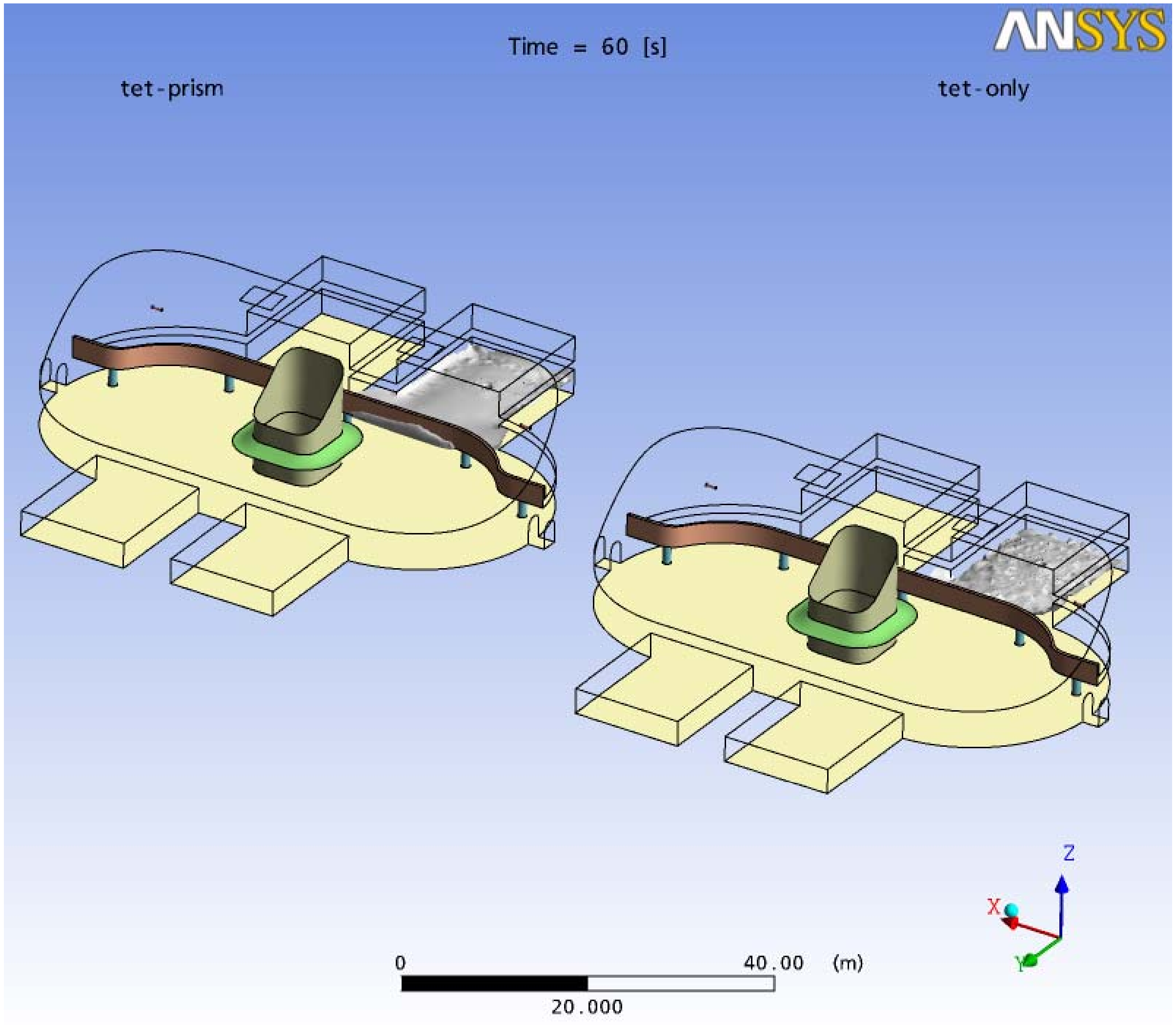




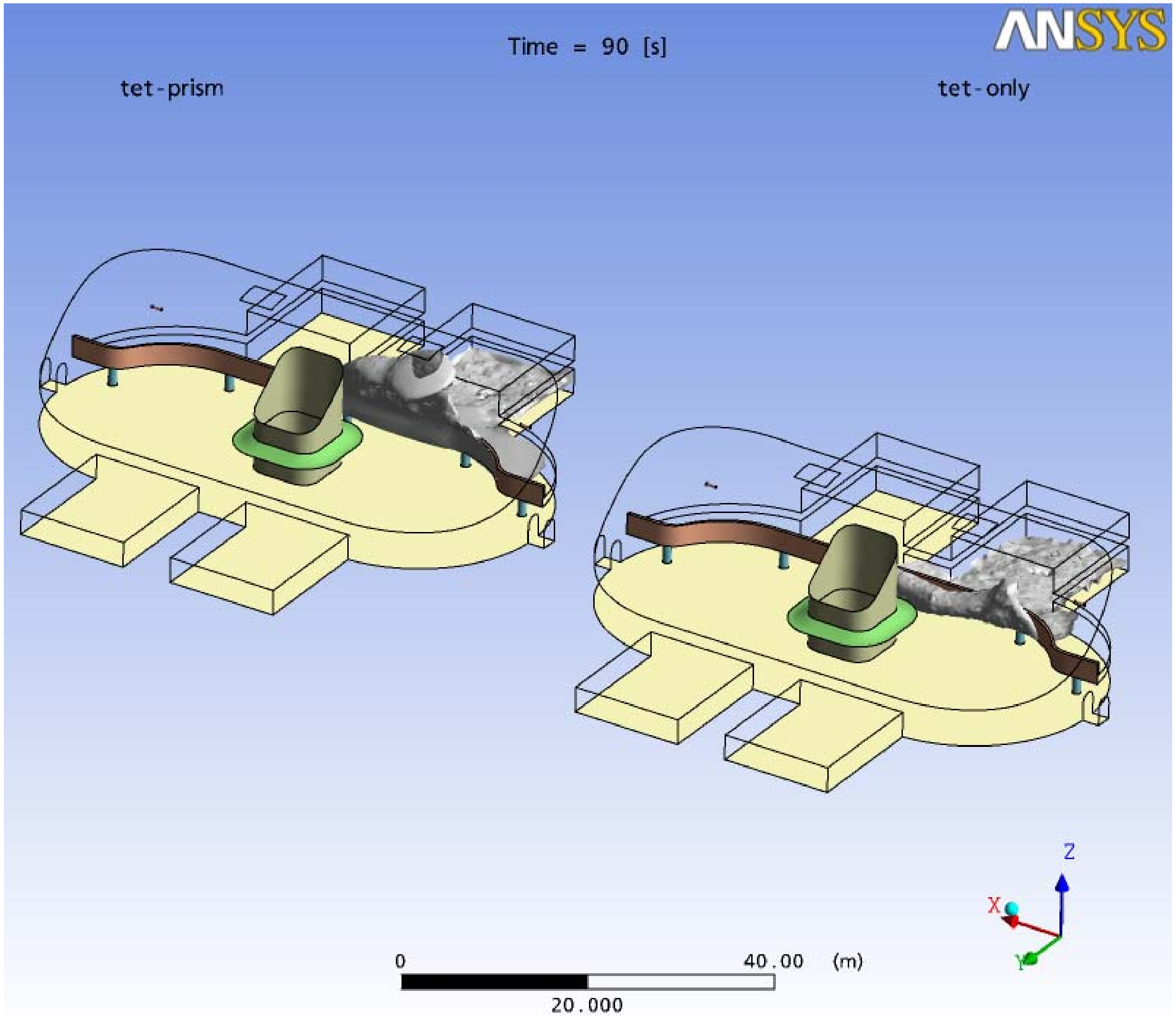
Use of inflation layers in ANSYS CFX



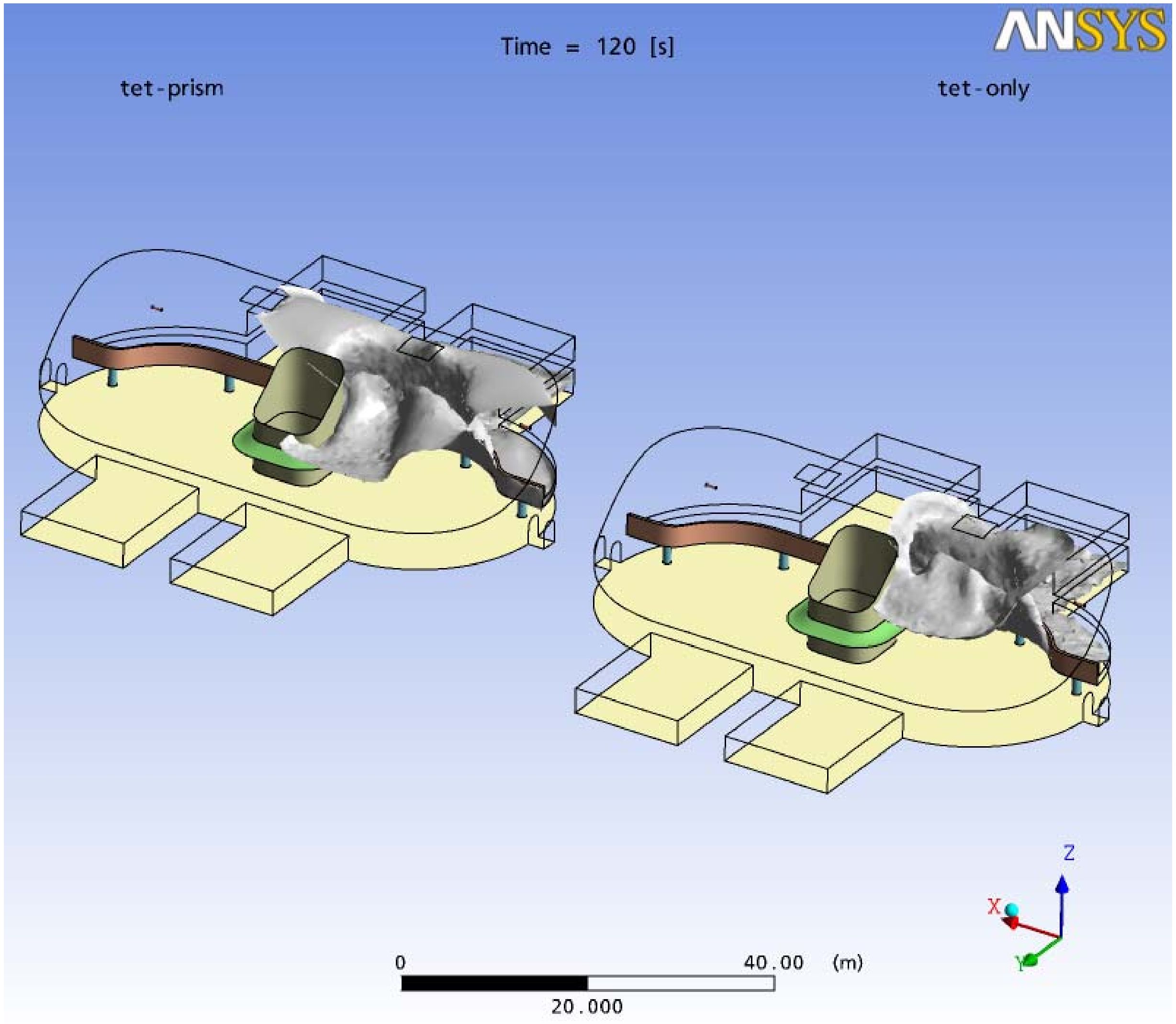
Effect of inflation layer on smoke spread



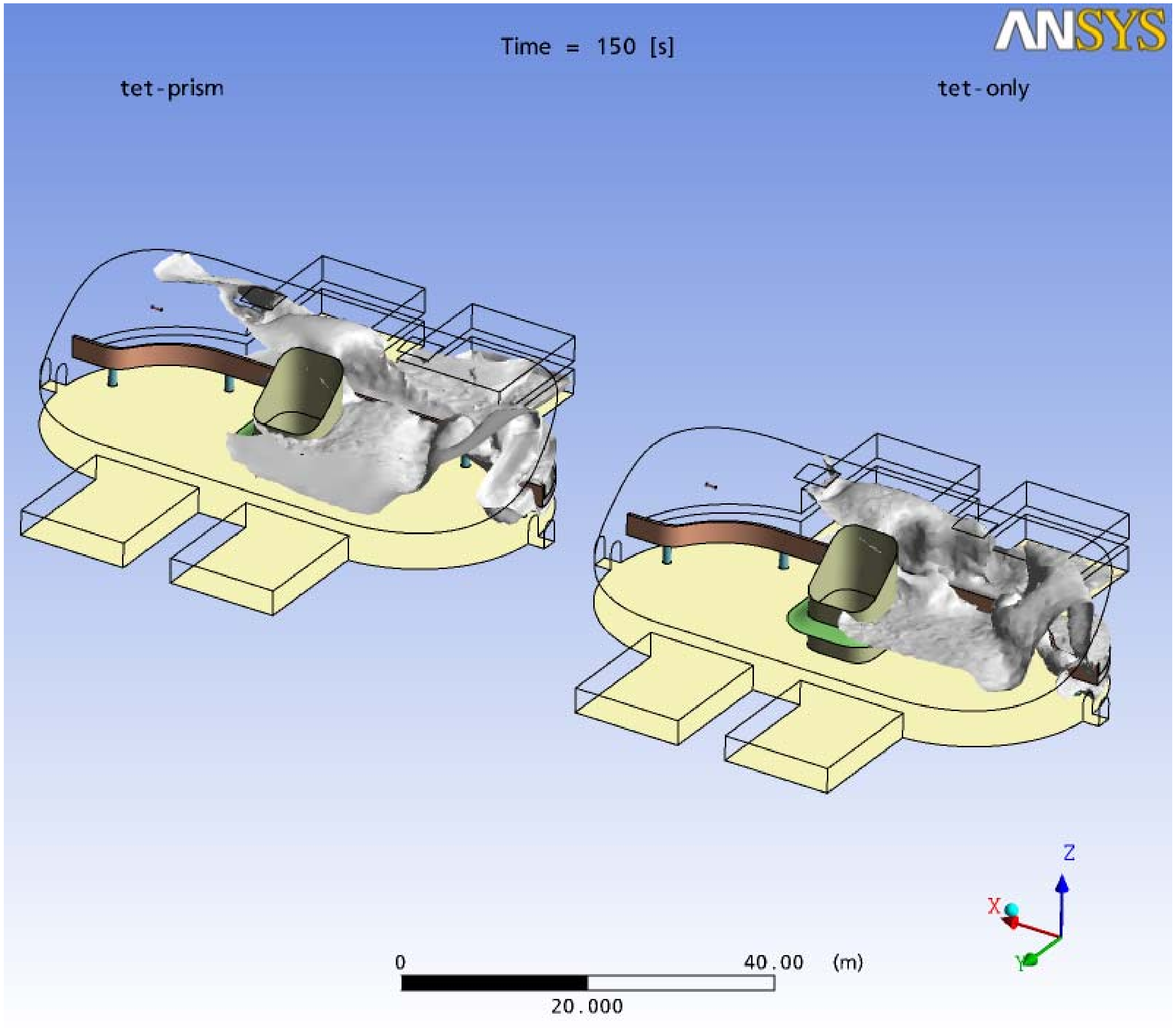
Effect of inflation layer on smoke spread



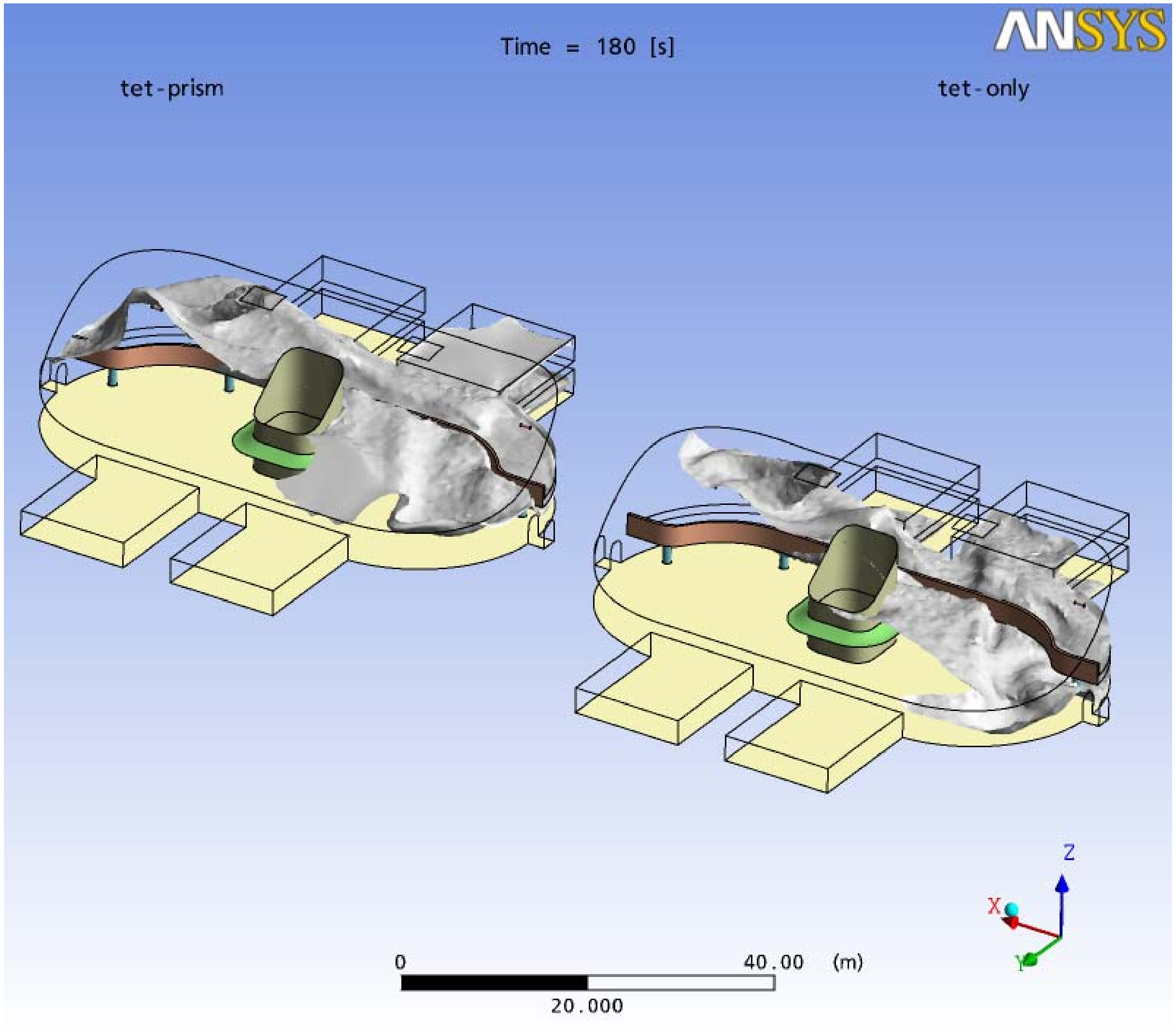
Effect of inflation layer on smoke spread



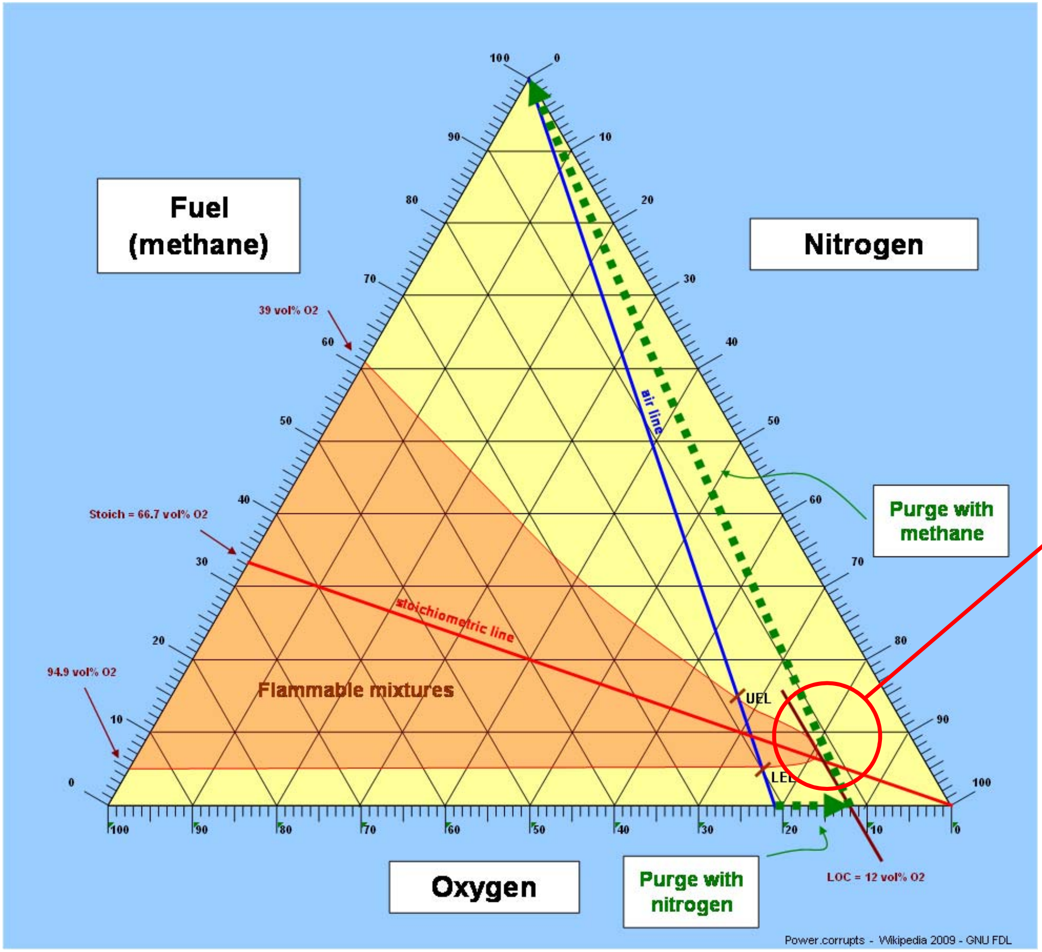
Effect of inflation layer on smoke spread



Effect of inflation layer on smoke spread



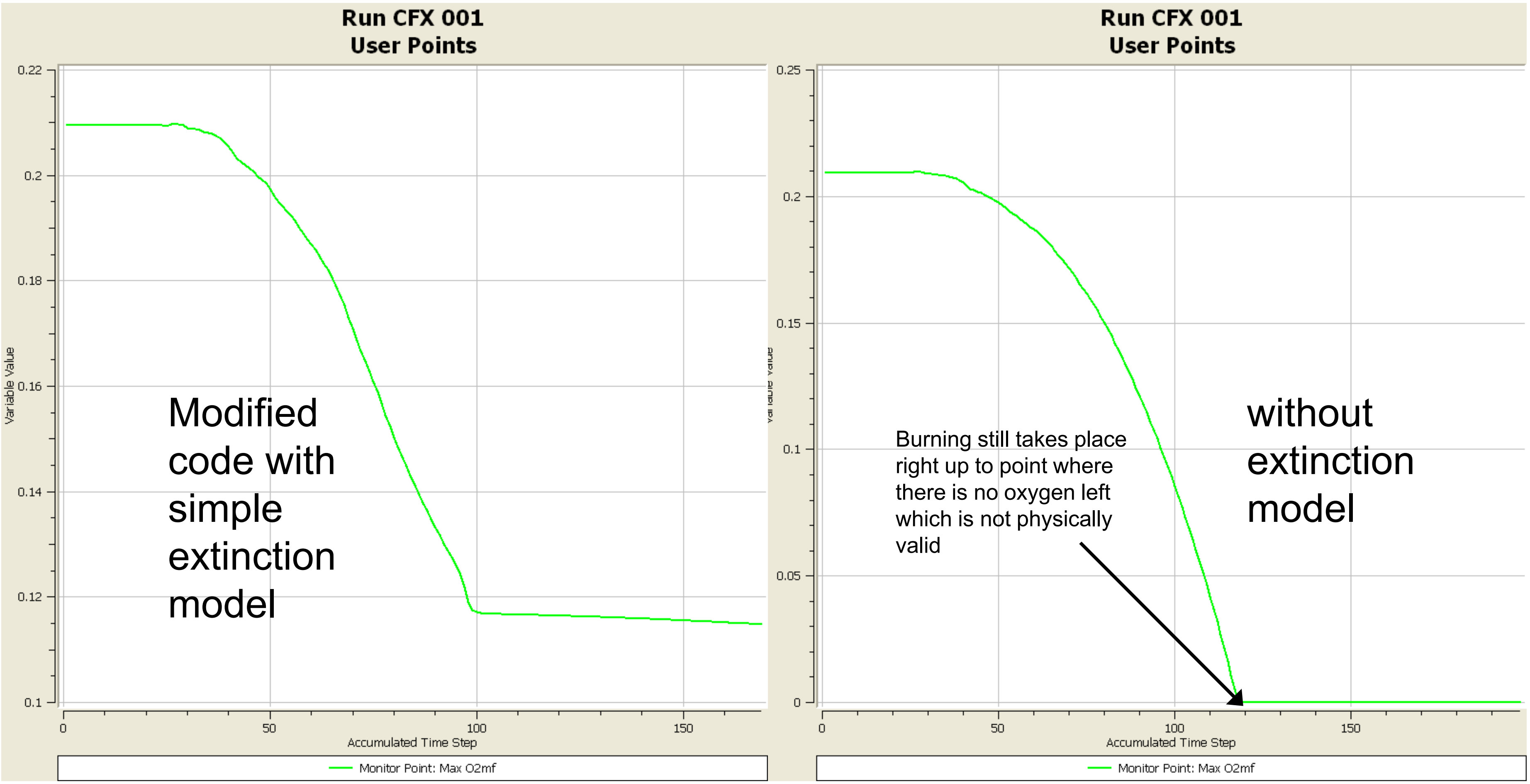
Reacting Models: Flammability



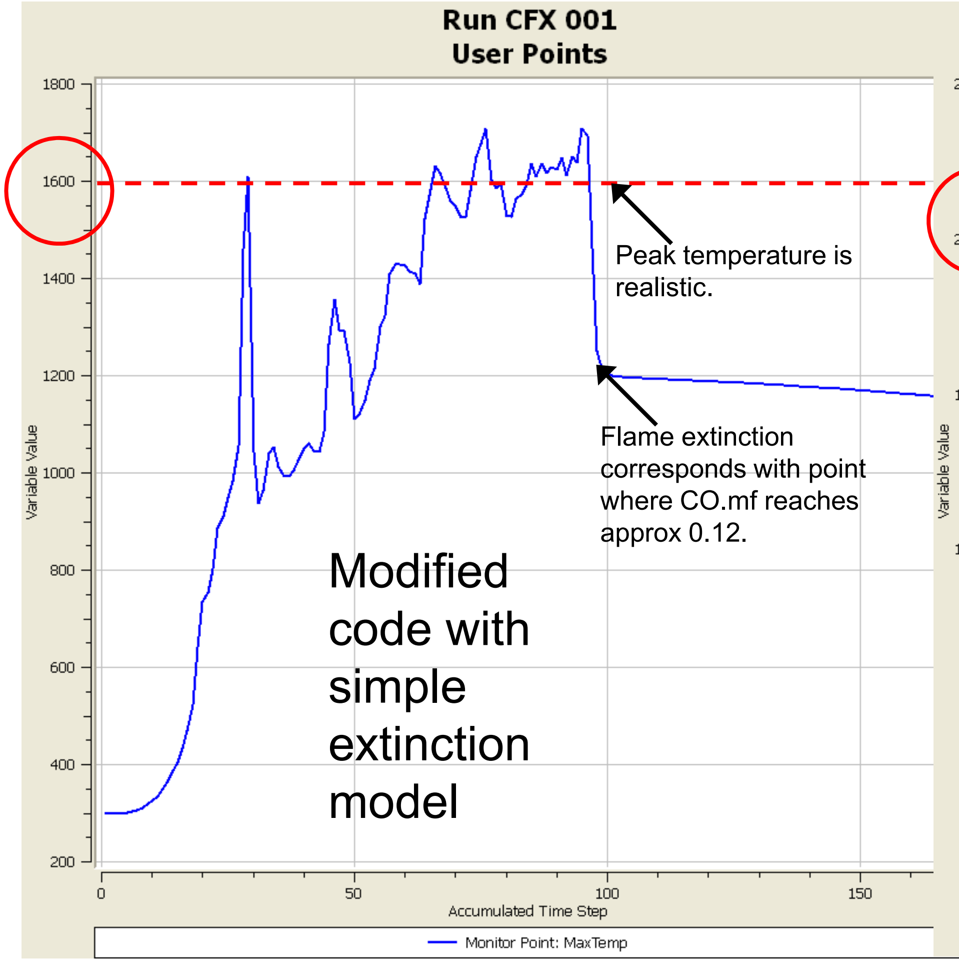
Mass fraction

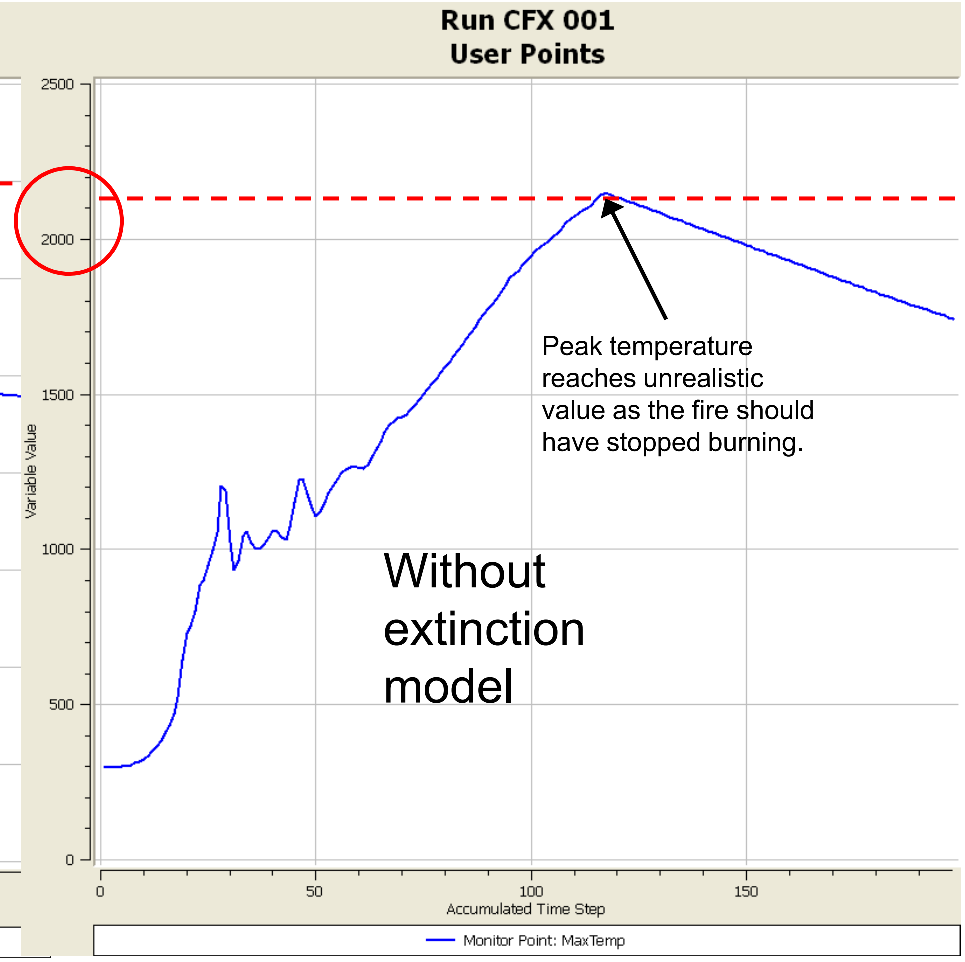
O2 = 0.12

Reacting Models: Comparisons – Oxygen mass fraction

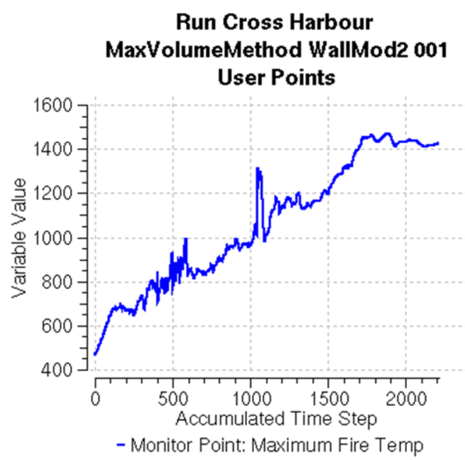


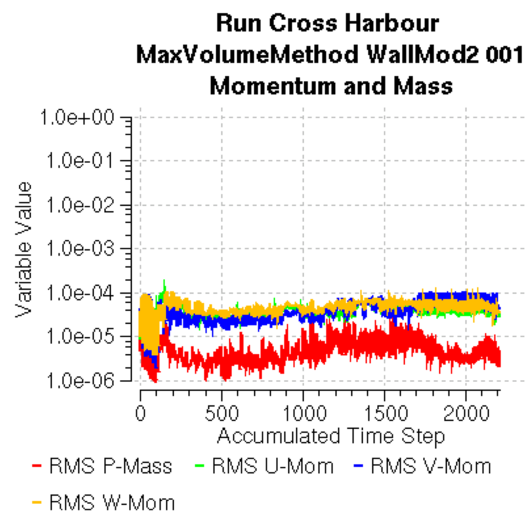
Comparisons - Temperature

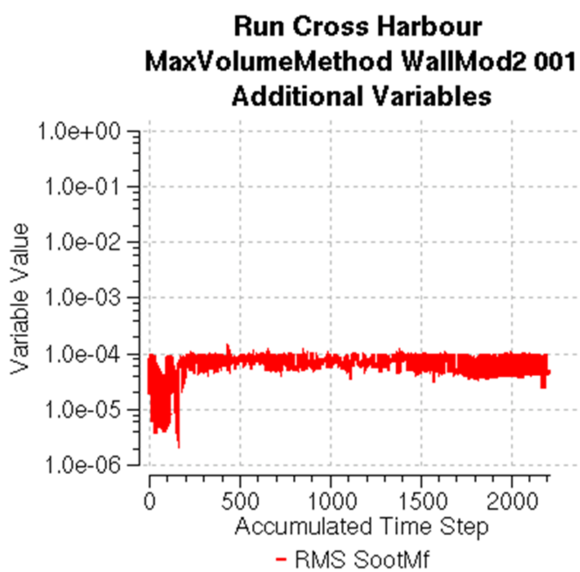


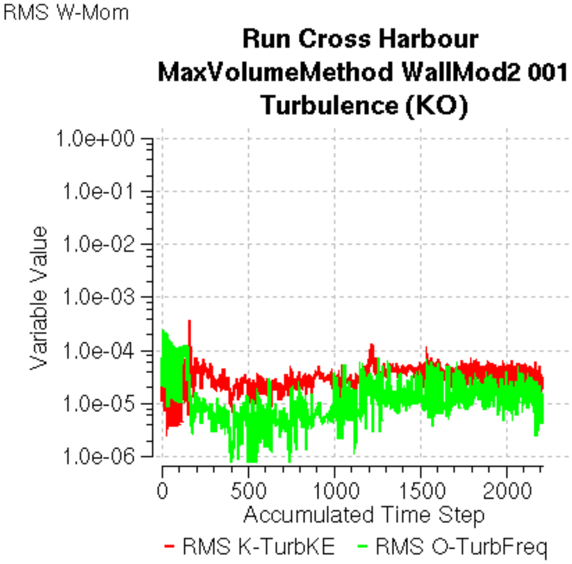


Example of convergence checks









Check list

1. The fuel area (or volume in the case of heat source method) should be sized to yield realistic average and peak temperatures. Maximum temperature should be shown in the report.

2. Maximum gas temperatures should be in the range 800 to 1300 C. Maximum temperatures should not exceed 1300 C.

3. An appropriate value for the Characteristic heat of combustion should be used. Usually between 24 to 26 MJ/Kg.

4. Soot yield should be specified in the report. 0.1 is normally specified for polyurethane fuel (FSS7.2 – 2.3).

5. Ceiling and outer walls should have heat transfer model applied rather than adiabatic (no heat loss assumption). Suggested value for heat transfer is 25kW/m^2/K (EN191-1-2).

6. Mass, momentum and energy conservation should be demonstrated.

7. Sensitivity to the mesh size chosen should be demonstrated.

8. Residual plots should be included (where available) to show that the time step value chosen is suitable. Transient time steps of typically 0.25 to 0.5 seconds are usually required to achieve reasonable convergence.