SECURE WITH STEEL

Training Course

Esch-sur-Alzette, 23-24 April 2018

Fire Dynamics Simulator (FDS)

The basics

Nicola Tondini, PhD

UNIVERSITY
OF TRENTO - Italy
Department of Civil, Environmental
and Mechanical Engineering

OBJECTIVES OF THE COURSE

- **To provide basic information about the use of FDS**
  1. How to build an FDS input file
  2. How to run FDS
  3. How to see and elaborate the outputs

- **The module aims to analyse case studies that can be relevant for structural fire engineering**
  - Evolution of the gas temperature in time and in space inside the compartment
  - Evolution of heat flux (or radiant intensities) in time and in space inside the compartment

OBJECTIVES OF THE COURSE

NET HEAT FLUX ACCORDING TO EN1991-1-2 (Eq. 3.1)

$$\dot{q}^{"}_{net} = \alpha_c \left(\theta_g - \theta_m\right) + \phi \sigma \varepsilon_f \varepsilon_m \left(\left(\theta_r + 273\right)^4 - \left(\theta_m + 273\right)^4\right)$$

$\dot{q}^{"}_{net}$ net heat flux received by a fire exposed surface (W/m^2)

α_c coefficient of heat transfer by convection (W/m^{2}K)

θ_g gas temperature close to the fire exposed member (°C)

θ_m surface temperature of the member (°C)

θ_r effective radiation temperature of the fire enviroment (°C)

ϕ configuration factor

ε_m surface emissivity of the member

ε_f emissivity of the fire

σ Stefan-Boltzmann constant (W/m^{2}K^4)

OBJECTIVES OF THE COURSE

NET RADIATIVE HEAT FLUX ACCORDING TO EN1991-1-2

Physically, the net heat flux to unit surface area due to radiation of Eq. (3.3) and the effective radiation temperature of the fire environment θ_r in Kelvin would be more precisely written

$$\dot{h}_{net,rad} = \varepsilon_m \sigma \left(\theta_r^4 - \left(\theta_m + 273\right)^4\right)$$

$$\theta_r^4 = \sum_i \varepsilon_{f.i} \phi_i \left(\theta_{f.i} + 273\right)^4$$

OBJECTIVES OF THE COURSE

NET HEAT FLUX ACCORDING TO EN1991-1-2 (Annex C)

$$\dot{q}''_{net} = \dot{q}'' - \alpha_c(\theta_m - 20) - \phi\sigma\varepsilon_f\varepsilon_m\left((\theta_m + 273)^4 - 293^4\right)$$

$\dot{q}''$ heat received by the fire exposed unit surface (W/m^2)

α_c coefficient of heat transfer by convection (W/m^{2}K)

θ_m surface temperature of the member (°C)

ϕ configuration factor

ε_m surface emissivity of the member

ε_f emissivity of the fire

σ Stefan-Boltzmann constant (W/m^{2}K^4)

ANALYSIS OF COMPARTMENT FIRES

3 problems have to be solved. Each of them is governed by different equations.

1. **Fire development** => Temperatures and flows in the compartment
2. **Thermal response** => Temperatures in the structural elements
   - Elements across the compartment.
   - Elements on the boundaries of the compartment.
3. **Mechanical response** => Behaviour of the structural elements.

WHAT IS TYPICALLY DONE IN STRUCTURAL FIRE ENGINEERING?

Sequential analysis

2. Thermal response

1. Fire development

3. Mechanical response

FUEL

OXYGEN

HEAT

Combustion of a carbon-based fuel material is an exothermic chemical reaction that implies the oxidation in water vapour and carbon dioxide

$$Fuel + O_2 \rightarrow H_2O + CO_2$$

For **fire** it is intended an uncontrolled and self-sustained combustion of fuel materials.

COMBUSTION TYPOLOGIES

1. **Diffusion flame** (It is a combustion process for which the fuel in its gaseous state and the oxygen converge into the reaction zone due to concentration differences Fick's law)

2. **Smouldering fire** (cigarette, charcoal)

3. **Spontaneous combustion** (hay)

4. **Premixed flame** (combustion engine)

FLAME

The flame is the volume part where the oxidation process occurs.

The combustion reaction that determines a flame is a phenomenon that happens at the gaseous state, also when the fuel material is liquid or solid.

FIRE

When the **fuel is liquid**, the change of state from liquid to gas occurs due to evaporative boiling at the surface.

Table 1.1 Properties of gaseous and liquid fuels[a]

	Formula	Melting point (°C)	Boiling point (°C)	Density (liq) (kg/m³)	Molecular weight
Hydrogen	H_2	−259.3	−252.8	70	2
Carbon monoxide	CO	−199	−191.5	422	28
Methane	CH_4	−182.5	−164	466	16
Ethane	C_2H_6	−183.3	−88.6	572	30
Propane	C_3H_8	−189.7	−42.1	585	44
n-Butane	n-C_4H_{10}	−138.4	−0.5	601	58
n-Pentane	n-C_5H_{12}	−130	36.1	626	72
n-Hexane	n-C_6H_{14}	−95	69.0	660	86
n-Heptane	n-C_7H_{16}	−90.6	98.4	684	100
n-Octane	n-C_8H_{18}	−56.8	125.7	703	114
iso-Octane[b]	iso-C_8H_{18}	−107.4	99.2	692	114
n-Nonane	n-C_9H_{20}	−51	150.8	718	128
n-Decane	n-$C_{10}H_{22}$	−29.7	174.1	730	142
Ethylene (ethene)	C_2H_4	−169.1	−103.7	(384)	28
Propene	C_3H_6	−185.2	−47.4	519	42
Acetylene (ethyne)	C_2H_2	−80.4	−84	621	26
Methanol	CH_3OH	−93.9	65.0	791	32
Ethanol	C_2H_5OH	−117.3	78.5	789	46
Acetone	$(CH_3)_2CO$	−95.3	56.2	790	58
Benzene	C_6H_6	5.5	80.1	874	78

[a] Data from Lide (1993/94).
[b] 2,2,4-Trimethyl pentane.

Drysdale, 1998

FIRE

When the **fuel is solid**, the change of state of solid to gas can occur in different modes. In most cases, a chemical decomposition of the fuel material happens and it is also called **pyrolysis**.

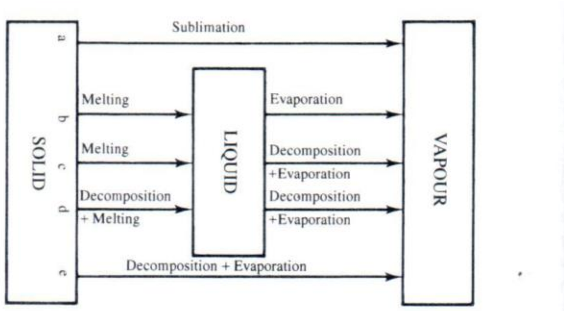


Figure 1.3 Different modes in which fuel vapour is generated from a solid (Table 1.3)

Drysdale, 1998

FIRE

DIFFUSION FLAME

A diffusion flame can be **turbulent or laminar** and **luminous or non-luminous**. In most cases the combustion of solids and liquids entails luminous flames, whose luminosity is given by the emissive effect of soot that forms within the flame.

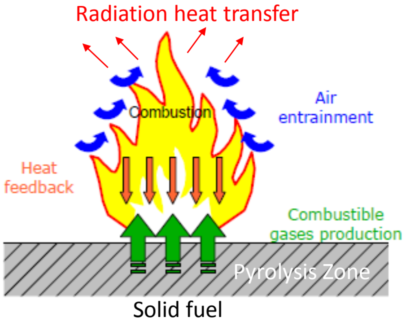
Fires are turbulent flows

FIRE

PHENOMENA INVOLVED

The physics behind a fire is complex with phenomena that are involved at different scales in time and in space:

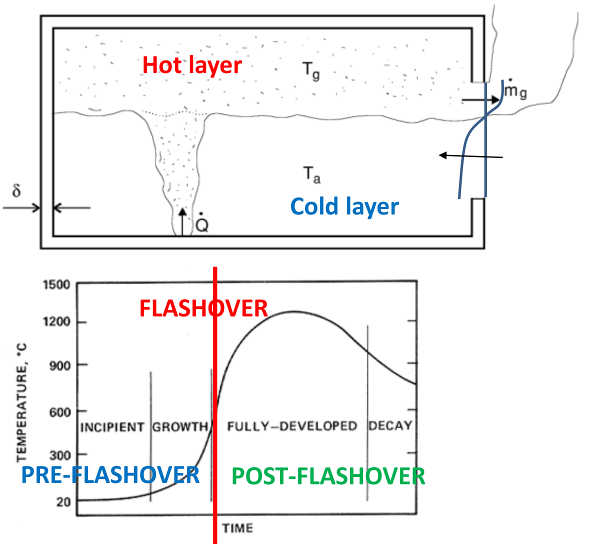
- Combustion
- Radiation
- Smoke movement and production
- Solid pyrolysis



Adapted from Desanghere, 2015

Full modelling is very demanding and simplifications are introduced

FIRE DEVELOPMENT INSIDE A COMPARTMENT



FORMULATION OF FUNDAMENTAL TRANSPORT EQUATIONS

They govern the fluid flow, heat transfer as well as turbulence.

- Equation of state
- Continuity equation (mass conservation)

The rate of increase of mass within the fluid element = the net rate at which mass enters the elemental volume

- Momentum equation

The rate of increase of momentum within the fluid element = the sum of forces acting on the fluid element

- Energy equation

The rate of increase of energy within the fluid element = the net rate of heat added to the fluid element + the net rate of work done on the fluid element + the rate of heat added or removed by the heat source on the fluid element

- Scalar equation (diffusion of gas concentration)

The rate of increase of scalar property of the fluid element within the fluid element = the net rate of scalar property added to the fluid element ± the rate of creation or destruction by external source on the fluid element

EQUATION OF STATE

Obtained through thermodynamics equilibrium. For a perfect or ideal gas

$$pV = nRT$$

p pressure (Pa)

V volume (m^3)

n number of moles

R universal gas constant (J mol^{-1}K^{-1})

T temperature (K)

The density of a gas can be determined

$$\rho = \frac{nM}{V} = \frac{pM}{RT}$$

ρ density (kg/m^3)

M molecular weight of a gas mixture (kg)

FIRE

EQUATION OF STATE

Flowing at high speed is characterized by a **compressible gas** and the density can change dramatically. Thus, the **equation of state** provides the **linkage** between the **energy conservation** equation and the equations for the **conservation of mass and momentum.**

For a **flowing gas at low-speed**, typical of burning fires, the gas behaves as **weakly compressible** fluid for which the density changes for temperature variations due to chemical reactions and unperturbed pressure.

$$\rho = \frac{p_0 M}{R_u T}$$

p_0 atmospheric pressure (Pa)

FIRE

EQUATIONS OF MOTIONS OF THE FLOW OF A COMPRESSIBLE NEWTONIAN FLUID IN A COMBUSTION FIRE SYSTEM

Mass

$$\frac{\partial \rho}{\partial t}+\frac{\partial(\rho u)}{\partial x}+\frac{\partial(\rho v)}{\partial y}+\frac{\partial(\rho w)}{\partial z}=0$$

$x-$Momentum

$$\frac{\partial(\rho u)}{\partial t}+\frac{\partial(\rho uu)}{\partial x}+\frac{\partial(\rho vu)}{\partial y}+\frac{\partial(\rho wu)}{\partial z}=\frac{\partial}{\partial x}\left[\mu\frac{\partial u}{\partial x}\right]+\frac{\partial}{\partial y}\left[\mu\frac{\partial u}{\partial y}\right]+\frac{\partial}{\partial z}\left[\mu\frac{\partial u}{\partial z}\right]-\frac{1}{\rho}\frac{\partial p}{\partial x}+$$

$$+\frac{\partial}{\partial x}\left[\mu\frac{\partial u}{\partial x}\right]+\frac{\partial}{\partial y}\left[\mu\frac{\partial v}{\partial x}\right]+\frac{\partial}{\partial z}\left[\mu\frac{\partial w}{\partial x}\right]+\frac{\partial}{\partial x}\left[\lambda\left(\frac{\partial u}{\partial x}+\frac{\partial v}{\partial y}+\frac{\partial w}{\partial z}\right)\right]+\sum F_x^{body\ forces}$$

Enthalpy

$$\frac{\partial(\rho h)}{\partial t}+\frac{\partial(\rho uh)}{\partial x}+\frac{\partial(\rho vh)}{\partial y}+\frac{\partial(\rho wh)}{\partial z}=\frac{\partial}{\partial x}\left[k\frac{\partial T}{\partial x}\right]+\frac{\partial}{\partial y}\left[k\frac{\partial T}{\partial y}\right]+\frac{\partial}{\partial z}\left[k\frac{\partial T}{\partial z}\right]+\dot{Q}_s$$

Scalar Property

$$\frac{\partial(\rho \varphi)}{\partial t}+\frac{\partial(\rho u\varphi)}{\partial x}+\frac{\partial(\rho v\varphi)}{\partial y}+\frac{\partial(\rho w\varphi)}{\partial z}=\frac{\partial}{\partial x}\left[\rho D\frac{\partial \varphi}{\partial x}\right]+\frac{\partial}{\partial y}\left[\rho D\frac{\partial \varphi}{\partial y}\right]+\frac{\partial}{\partial z}\left[\rho D\frac{\partial \varphi}{\partial y}\right]+\dot{R}_s$$

ρ density, (u,v,w) flow field, p pressure, μ dynamic viscosity, λ second viscosity, h enthalpy

T temperature, k thermal conductivity, $\dot{Q}_s$ rate of heat added or removed by heat source on the fluid

φ scalar (concentration), D mass diffusivity, $\dot{R}_s$ increase or decrease of the scalar property due to some prescribed external sources

HEAT TRANSFER MECHANISMS

CONDUCTION

Heat transfer in solids. It is very weak in gases.

The **Fourier's law** reads

$$q_i=-k_{ij}T_{,j}$$
$$\boldsymbol{q}=-\boldsymbol{k}\nabla T$$

q_i *is the heat flux vector across a unit surface*

k_{ij} *is the thermal conductivity for an anisotropic material*

Heat conduction equation for an homogeneous material

$$kT_{,ii}=c\rho\dot{T}-R$$
$$k\left(\frac{\partial^2 T}{\partial x^2}+\frac{\partial^2 T}{\partial y^2}+\frac{\partial^2 T}{\partial z^2}\right)=c\rho\frac{\partial T}{\partial t}-R$$

HEAT TRANSFER MECHANISMS

CONVECTION

Heat transfer due to motion of a fluid, e.g. hair dryer. The convective flux is

$$\dot{q}'' = h\left(T_g - T_m\right)$$

h is the convection coefficient

T_g is the gas (or fluid) temperature

T_m is the material temperature

It can be **natural** (governed by density difference) or **forced** (governed by forced flow around the solid).

HEAT TRANSFER MECHANISMS

RADIATION

Heat transfer due to electromagnetic waves. No medium is needed.

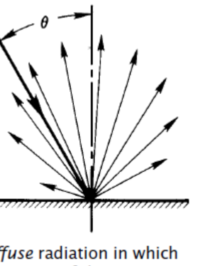
- Radiation intensity
- Wavelength

Blackbody is a body that absorbs all incoming radiation

$$\dot{q}'' = \sigma T_m^4$$

σ Stefan-Boltzmann constant

T_m material surface temperature



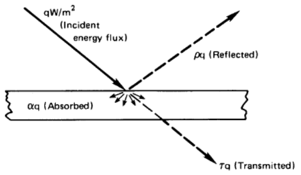
Diffuse radiation in which directions of departure are uninfluenced by incoming ray angle, θ.

HEAT TRANSFER MECHANISMS

RADIATION

Emissivity is the ratio between the radiation emitted by a material and a blackbody at a given temperature. In general, it depends on the wavelength. The radiative heat flux of a gray body is

$$\dot{q}'' = \frac{e(\lambda)}{e_b(\lambda)} = \varepsilon(\lambda)\sigma T_m^4 = \varepsilon\sigma T_m^4$$

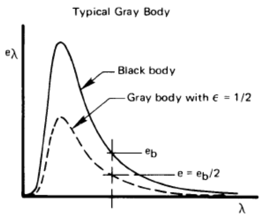


e radiation emitted by a material at T_m

e_b radiation emitted by a blackbody at T_m

ε emissivity [0,1]

λ wavelength



HEAT TRANSFER MECHANISMS

RADIATION

Soot is the primary emitter and absorber of thermal radiation. Thus, it provides the mechanism of radiative heat loss (air is transparent).

It absorbs and diffuses the incident radiation

it is a participating medium

ANALYSIS OF COMPARTMENT FIRES

FIRE DEVELOPMENT (EN1991-1-2)

1. **Nominal temperature-time curves** (e.g. ISO 834)
2. **Natural Fire Safety Concept**
   - Simplified models
     - Parametric temperature-time curves (Annex A)
     - Localised fires (Annex C)
   - Advanced models
     - Zone models (Annex D)
     - Computational Fluid Dynamics (CFD)

CFD OVERVIEW

WHAT IS CFD?

- CFD is the study of fluid flowing that can change in time and in space.
- The fluid motion is described by mathematical equations that are translated into numerical (algebraic) equations.
- The algebraic equations are then solved with dedicated techniques.

Thus, CFD is an ensemble of three disciplines:

1. Fluid mechanics
2. Mathematics
3. Computer science

RECOMMENDED READINGS

CFD

- *Richard H. Pletcher, John C. Tannehill and Dale Anderson, **Computational Fluid Mechanics and Heat Transfer**, 3rd Edition*
- *Guan Heng Yeoh and Kwok Kit Yue, **Computational Fluid Dynamics in Fire Engineering: Theory, Modelling and Practice**, 1st Edition*

Fire Dynamics

- *Drysdale D., **An introduction to Fire dynamics** 3rd Edition, Wiley, Chichester UK.*
- *Society of Protection Fire Engineers, **The SFPE Handbook of Fire Protection Engineering**, NFPA, Quincy, USA.*

CFD OVERVIEW

BRIEF CHRONOLOGY

1950s: First developments.

1980s: First applications of large extent began with the improvement of computer capabilities.

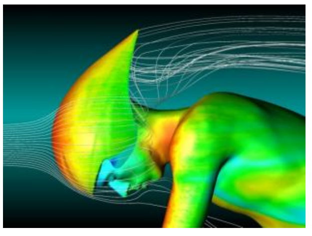
2000s: Widely used in the industry with a huge market behind it.

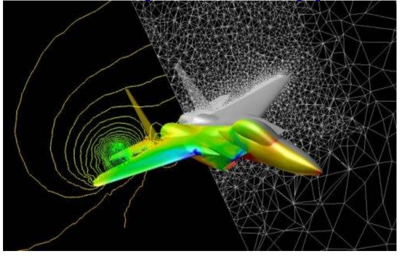
SOFTWARE

For fire modelling: FDS, JASMINE, SOFIE, FLUENT, ANSYS CFX, STAR-CCM+

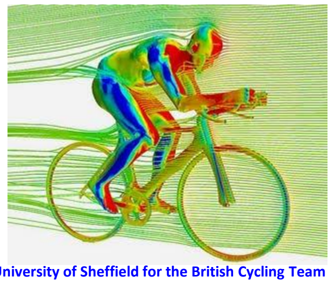
CFD OVERVIEW

COMPUTATIONAL FLUID DYNAMICS (CFD) analyses intend to simulate any type of fluid flow in many various applications.

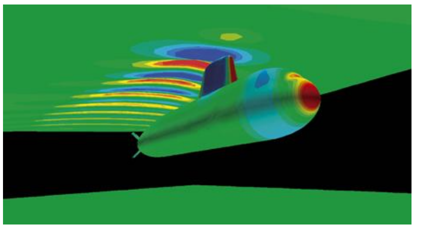




Cobalt Solutions and Arizona State University



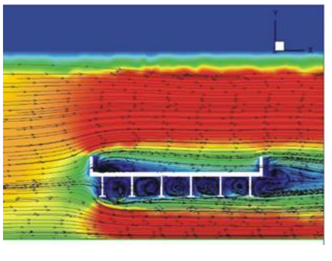
University of Sheffield for the British Cycling Team

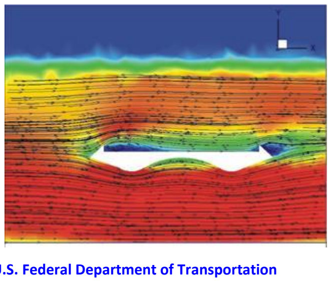


SSPA Sweden AB

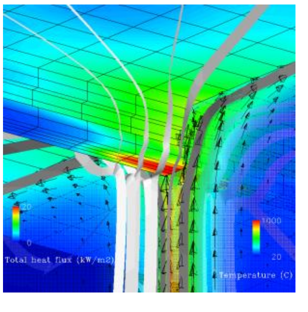
CFD OVERVIEW

Civil engineering applications





U.S. Federal Department of Transportation



Dr. S. Welch - University of Edinburgh

CONCEPTUAL STEPS IN MODELLING

- **Real phenomenon** that we want to model
- **Physical model** that involves the understanding of the phenomenon and the choice of the laws that describe it.
- **Mathematical model** that implies the choice of the numerical method to solve.
- **Computer model**: the implementation and resolution of the problem.

  GARBAGE IN => GARBAGE OUT

FIRE DYNAMICS SIMULATOR

- Developed by the **National Institute of Standards and Technology** (NIST). First release in 2000.

- **Free software** that can be downloaded at

  https://pages.nist.gov/fds-smv/downloads.html

  Good forum where to post questions

  https://groups.google.com/forum/#!forum/fds-smv

- It is a CFD code used to model fire-driven fluid flow.

FIRE DYNAMICS SIMULATOR

- **Hydrodynamic model**
  - FDS solves numerically a form of the Navier-Stokes equations appropriate for low-speed (Mach < 0.3), thermally driven flow with emphasis on smoke and heat transport fires.
- **Combustion model**
  - FDS, by default, uses a single step, mixing controlled chemical reaction.
- **Radiation transport**
  - Radiative heat transfer is included in the model via the solution of the radiation transport equation for a gray gas.

FIRE DYNAMICS SIMULATOR

HYDRODYNAMIC MODEL

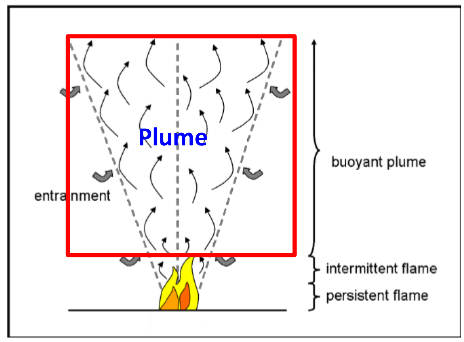
By default, **turbulence** is treated by means of **LARGE EDDY SIMULATION (LES)**. That is, the turbulent reacting flow is characterized by solving the macroscopic large-scale motion, e.g. large eddies, through the governing equations, while the microscopic small-scale motion, e.g. diffusion, is approximated by **subgrid scale (SGS) modelling**.

Small-scale phenomena: the combustion zone above the fuel represents a region where the local mixing of gasified fuel and air reacts to produce combustion products

Large-scale phenomena: the combustion zone represents a source of buoyancy, which induces large-scale mixing of air and combustion products, forming a plume.

HYDRODYNAMIC MODEL

Small-scale vs. Large-scale phenomena

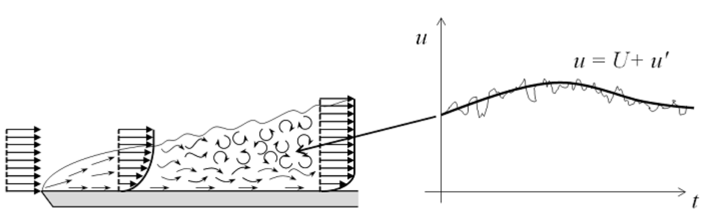


NOTE: LES is not symmetric even if the problem is symmetric. Be careful if you intend to use the MIRROR command.

OTHER METHODS TO RESOLVE TURBULENCE

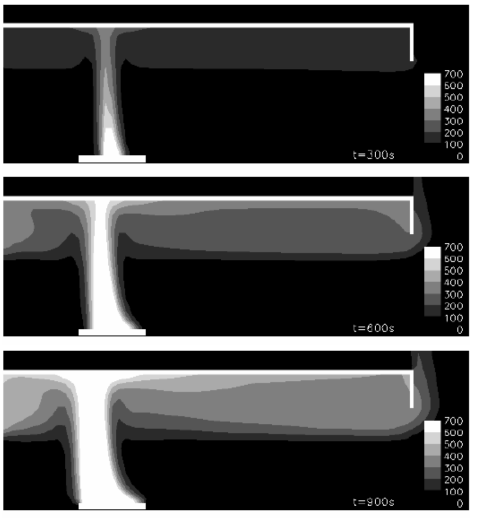
- **Direct Numerical Simulation (DNS)**. Modelling all scales exactly. Extremely demanding. FDS can exploit it.

- **Reynolds Average Navier Stokes** (RANS). Approximation by time-averaged the Navier-Stokes equations (e.g. JASMINE, SOFIE).



OTHER METHODS TO RESOLVE TURBULENCE

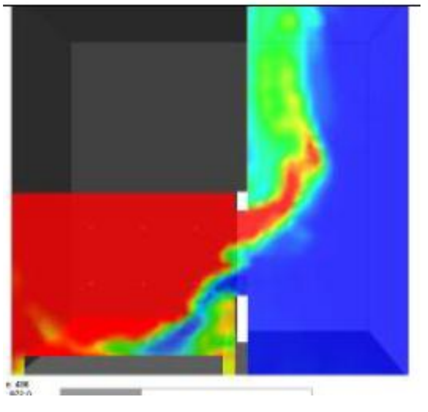
Typical RANS solution. The contour lines are regular.

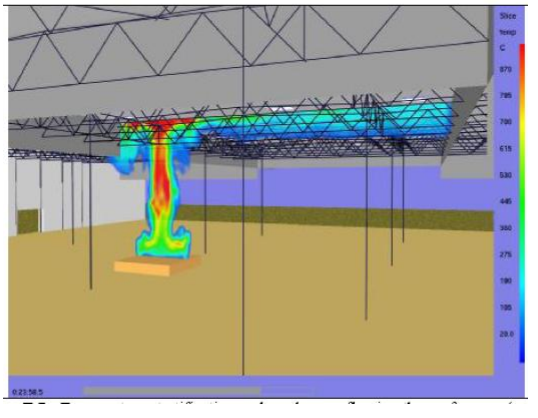


FIRE DYNAMICS SIMULATOR

OTHER METHODS TO RESOLVE TURBULENCE

Typical LES solution. The eddies are visible.





FIRESTRUCT (2003)

The mesh has to be suitable to resolve the eddies.

TURBULENT VISCOSITY MODELS

In **LES** approach the **small dissipative scales are not solved accurately**. So, the SGS models aim to represent the kinetic losses due to viscous forces and they attempt to produce the effect of SGS stresses in statistical sense.

- Deardoff (Default)
- Dynamic Smagorinsky
- Vreman
- Constant Smagorinsky
- Renormalization Group (RNG) Model

FIRE DYNAMICS SIMULATOR

COMBUSTION MODEL

FDS uses a three **lumped species** approach, which means that the lumped species are the **air, fuel and products**. Fuel and products are explicitly computed => ONE CHEMICAL REACTION

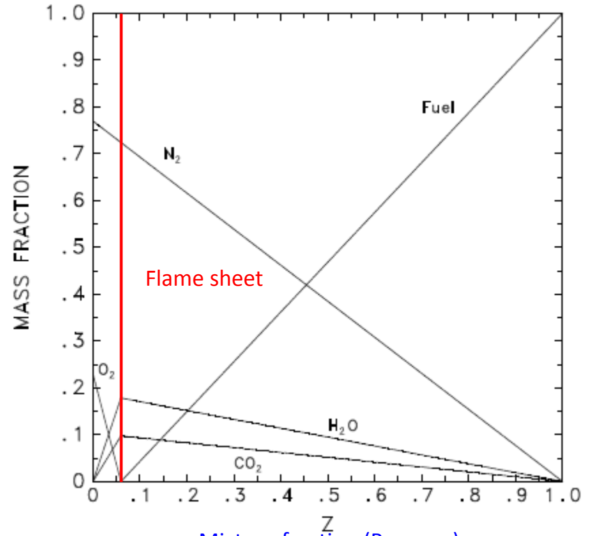
Non-premixed fuel and oxidant concentrations can be derived in the limit of fast chemistry. That is

For **single-step, irreversible and infinitely fast chemical reaction** the main assumption is:

- The reaction is so fast that the fuel and oxidant cannot co-exist everywhere except within an infinitely thin flame sheet.

FIRE DYNAMICS SIMULATOR

COMBUSTION MODEL



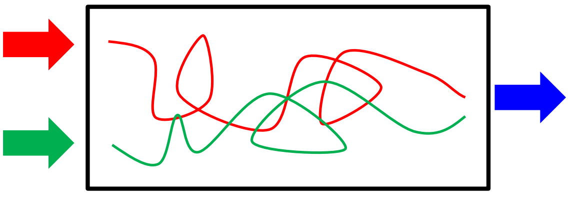
Mixture fraction (Propane)

McGrattan et al. (2001)

FIRE DYNAMICS SIMULATOR

COMBUSTION MODEL

The mixture fraction is a conserved scalar quantity used in the analysis of non-premixed combustion to describe the degree of scalar mixing between the fuel and the oxidant.



COMBUSTION CHAMBER

FIRE DYNAMICS SIMULATOR

RADIATION TRANSPORT

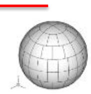
The **Radiation Transport Equation (RTE)** is solved according to finite volume method for a **finite number of angles**. Default number 100.

While FDS does have an option to divide the radiation spectrum into a relatively small number of bands and solve a separate RTE for each band, it is usually not necessary because **in real fires soot is the dominant source and sink of thermal radiation and is not particularly sensitive to wavelength**.

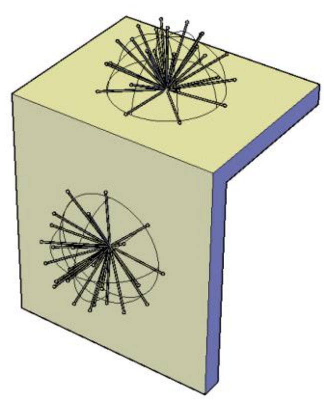
The **mean absorption coefficient** is a function of species composition and temperature.

FIRE DYNAMICS SIMULATOR

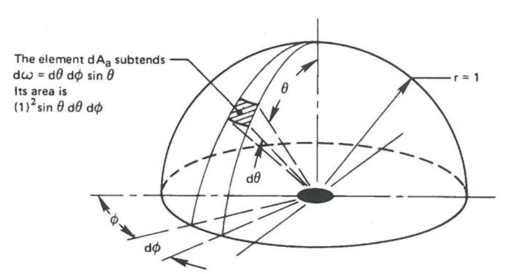
RADIATION TRANSPORT



Integration of the radiation intensities for a **finite number of angles**.



Flux received by a surface



$$dq = I d\omega \cos\vartheta \text{ where } d\omega = \sin\vartheta d\vartheta d\varphi$$

$$q = \int_0^{2\pi}\int_0^{\pi/2} I \sin\vartheta \cos\vartheta d\vartheta d\varphi$$

RADIATION TRANSPORT

Another important parameter is the **Radiative Fraction**. It is a function of both the flame temperature and chemical composition, neither of which are reliably calculated in a large scale fire calculation because the flame sheet is not well-resolved on a relatively coarse numerical grid.

Radiative Fraction specifies explicitly the fraction of the total combustion energy that is released in the form of thermal radiation.

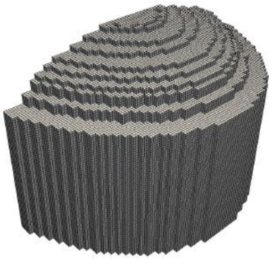
Default value is 0.35.

MESH

FDS uses a structured rectilinear mesh. Division in cells.

- **Scalar quantities** are assigned to the center of each grid cell
- **Vector components** are assigned at the appropriate cell faces

Staggered grid



NUMERICAL INTEGRATION OF EQUATIONS

FLUID FLOW: The governing equations that are partial difference equations are converted into algebraic equations that are solved according to the finite difference method

COMBUSTION: One chemical reaction based on mixture fraction

RADIATION: the Radiation Transport Equation is solved according to finite volume method. Absorption coefficient is related to gas composition.

STABILITY OF THE INTEGRATION SCHEME

The Courant-Friedrichs-Lewy (CFL) constraint

$$CFL = \delta t \frac{\|\mathbf{u}\|}{\delta x} \overset{default}{=} \delta t \max\left(\frac{|u|}{\delta x}, \frac{|v|}{\delta y}, \frac{|w|}{\delta z} \right) \approx 1$$

δt time step

$\|\mathbf{u}\|$ norm of the flow field

δx mesh size along one direction

Physically, the constraint says that a fluid element should not traverse more than one cell within a time step.

FIRE DYNAMICS SIMULATOR

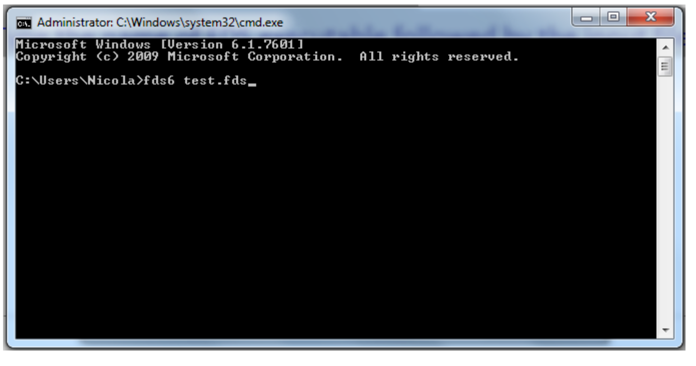
USING FDS: WHAT YOU NEED

- The **FDS executable** to run the analysis *.exe
- A **text editor** to write the input file *.fds and read output files *.out
- **EXCEL** to read and elaborate the data *.csv
- **SMOKEVIEW** to visualize the results of the analysis
- The **User's Guide** and for deepening the **Technical Reference**

FIRE DYNAMICS SIMULATOR

RUNNING FDS

- Create a folder where the FDS executable shall be copied in
- Open a command prompt windows
- Type the name of the FDS executable followed by the input file name



RUNNING FDS

In order to speed up the analysis FDS can be run with **MPI Parallel Processing**, i.e. on different PCs, and/or with **OpenMP** to exploit the presence of multiple cores.

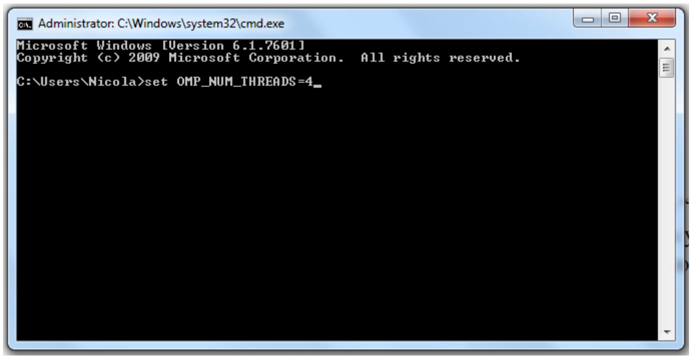
- **MPI**: the domain is divided into different parts to be run in parallel on multiple machines
- **OpenMP**: allows a single computer to run a single or multiple mesh FDS simulation on multiple cores

FIRE DYNAMICS SIMULATOR

RUNNING FDS: OpenMP

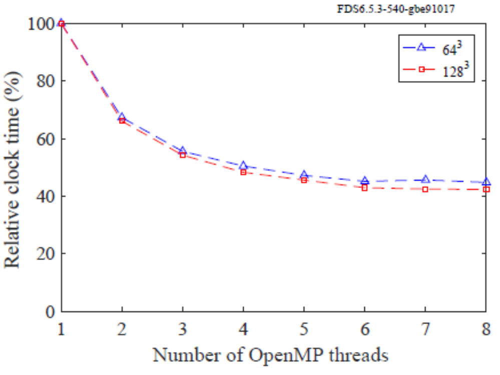
In order to change the number of cores (default 4 cores) in the command prompt one should type

```
set OMP_NUM_THREADS=4
```



RUNNING FDS: OpenMP

The multi-core option improves the speed up to 4 cores (increase by 2)



If multiple fire scenarios are to be analysed, they can be run simultaneously by using the OpenMP option

WRITING AN FDS INPUT FILE: File structure

The input file starts with namelist `&HEAD` and end with `&TAIL`

- Each namelist starts with **&** followed by **4 characters**, e.g. `&HEAD`
- Parameters are given by leaving blanks. (Suggestion: change only the parameters that you want to use with a different default value)
- Each namelist terminates with a forward slash **/**

EXAMPLE

```
&HEAD CHID='radiation_box', TITLE=‘Wall heat transfer in a box'/
```

`CHID` is a string that tags the output files (max 30 characters)

`TITLE` is a string that describes the analysis (max 60 characters)

FIRE DYNAMICS SIMULATOR

WRITING AN FDS INPUT FILE: File structure

Comments

Anything outside **&** and **/** is considered as a comment

EXAMPLE

```
Radiation inside a box
======================
&HEAD CHID='radiation_box', TITLE='Test wall heat transfer in a box'/
```

← Comments

FIRE DYNAMICS SIMULATOR

WRITING AN FDS INPUT FILE: Namelist

Group Name	Namelist Group Description
BNDF	Boundary File Output
CLIP	Clipping Parameters
CSVF	Velocity Input File
CTRL	Control Function Parameters
DEVC	Device Parameters
DUMP	Output Parameters
HEAD	Input File Header
HOLE	Obstruction Cutout
HVAC	Heating, Vent., Air Cond.
INIT	Initial Condition
ISOF	Isosurface File Output
MATL	Material Property

FIRE DYNAMICS SIMULATOR

WRITING AN FDS INPUT FILE: Namelist

MESH	Mesh Parameters
MISC	Miscellaneous
MULT	Multiplier Parameters
OBST	Obstruction
PART	Lagrangian Particle
PRES	Pressure Solver Parameters
PROF	Profile Output
PROP	Device Property
RADI	Radiation
RAMP	Ramp Profile
REAC	Reaction Parameters
SLCF	Slice File Output
SPEC	Species Parameters
SURF	Surface Properties
TABL	Tabulated Particle Data
TIME	Simulation Time
TRNX	Mesh Stretching
VENT	Vent Parameters
ZONE	Pressure Zone Parameters

FIRE DYNAMICS SIMULATOR

WRITING AN FDS INPUT FILE: File structure

```
&HEAD CHID='WTC_05', TITLE='WTC Phase 1, Test 5' /
&MESH IJK=90,36,38, XB=-1.0,8.0,-1.8,1.8,0.0,3.82 /
&TIME T_END=5400. /
&MISC TMPA=20./
&DUMP NFRAMES=1800, DT_HRR=10., DT_DEVC=10. /
&REAC FUEL = 'N-HEPTANE'
         C = 7.
         H = 16.
SOOT_YIELD = 0.015 /
&OBST XB= 3.5, 4.5,-1.0, 1.0, 0.0, 0.0, SURF_ID='STEEL FLANGE' / Fire Pan
...
&SURF ID  = 'STEEL FLANGE'
    COLOR = 'BLACK'
  MATL_ID = 'STEEL'
  BACKING = 'EXPOSED'
THICKNESS = 0.0063 /
...
&VENT MB='XMIN', SURF_ID='OPEN' /
...
&SLCF PBY=0.0, QUANTITY='TEMPERATURE', VECTOR=.TRUE. /
&BNDF QUANTITY='GAUGE HEAT FLUX' /
&DEVC XYZ=6.04,0.28,3.65, QUANTITY='VOLUME FRACTION', SPEC_ID='OXYGEN', ID='EO2_FDS' /
&TAIL / End of file.
```

FIRE DYNAMICS SIMULATOR

WRITING AN FDS INPUT FILE: &MESH

```
&MESH IJK=10,20,30, XB=0.0,1.0,0.0,2.0,0.0,3.0 /
```

`IJK` divisions in cells in the three directions
`XB` defines the volume with a sextuplet

It is a volume starting from (x,y,z) = (0,0,0) of sides 1 m x 2 m x 3 m

It is best if the mesh cells resemble cubes. In the example 10 cm.

FIRE DYNAMICS SIMULATOR

WRITING AN FDS INPUT FILE: &MESH

The pressure solver in FDS employs Fast Fourier Transforms (FFTs) in the **y and z directions**, and this algorithm works most efficiently if the number of cells in these directions (the `J` and `K` of `IJK`) **can be factored into low primes, like 2, 3, and 5**.

List of numbers between 1 and 1024 that can be factored

2	3	4	5	6	8	9	10	12	15	16	18	20	24	25
27	30	32	36	40	45	48	50	54	60	64	72	75	80	81
90	96	100	108	120	125	128	135	144	150	160	162	180	192	200
216	225	240	243	250	256	270	288	300	320	324	360	375	384	400
405	432	450	480	486	500	512	540	576	600	625	640	648	675	720
729	750	768	800	810	864	900	960	972	1000	1024				

WRITING AN FDS INPUT FILE: &MISC

Miscellaneous parameters

```
&MISC  TMPA = 25.
       TURBULENCE_MODEL = 'CONSTANT SMAGORINSKY'/
```

`TMPA` sets the ambient temperature (default 20°C)

`TURBULENCE_MODEL` sets the turbulence model (default DEARDORFF)

WRITING AN FDS INPUT FILE: &TIME

Time parameters

```
&TIME  T_BEGIN = 0., T_END=5400. /
```

`T_BEGIN`	starting time of the simulation	(default 0 s)
`T_END`	ending time of the simulation	(default 1 s)
`DT`	time step	(adjusted to comply with CFL)

$$DT \overset{default}{=} \frac{5\sqrt[3]{\delta x \delta y \delta z}}{\sqrt{gH}}$$

$\delta x, \delta y, \delta z$ dimensions of the smallest mesh cell

g gravity acceleration

H is the height of the computational domain

WRITING AN FDS INPUT FILE: &DUMP

It is used to manage the outputs

```
&DUMP NFRAMES=1800, DT_HRR=10., DT_DEVC=10./
```

NFRAMES	number of frame of output data (default 1000)	
DT_HRR	heat release rate interval	(T_END-T_BEGIN)/NFRAMES
DT_DEVC	device output dump interval	(T_END-T_BEGIN)/NFRAMES

WRITING AN FDS INPUT FILE: &REAC

It is used to define the (one) chemical reaction. Simple chemistry considers a single fuel species that is composed primarily of C, H, O and N that reacts with oxygen in one mixing controlled step to form H_2O, CO_2, soot and CO.

FDS has already a number of fuels in its library

```
&REAC          FUEL = 'N-HEPTANE'
                  C = 7.
                  H = 16.
         SOOT_YIELD = 0.037
           CO_YIELD = 0.01
HEAT_OF_COMBUSTION = 44600. /
```

WRITING AN FDS INPUT FILE: &REAC

`FUEL` specifies the type of fuel

`C,H,O,N` defines the fuel formula

`SOOT_YIELD` defines the fraction of fuel mass converted in soot (default 0)

`CO_YIELD` defines the fraction of fuel mass converted in CO (default 0)

`HEAT_OF_COMBUSTION` defines the heat of combustion (kJ/kg)

WRITING AN FDS INPUT FILE: &REAC

Species	Mol. Wt. (g/mol)	Formula	σ (Å)	ε/k (K)	Liquid	RadCal Surrogate
ACETONE	58.07914	C_3H_6O	4.6	560.2	Y	MMA
ACETYLENE	26.037280	C_2H_2	4.033	231.8		PROPYLENE
ACROLEIN	56.063260	C_3H_4O	4.549	576.7	Y	MMA
AMMONIA	17.03052	NH_3	2.9	558.3	Y	
ARGON	39.948000	Ar	3.42	124.0	Y	
BENZENE	78.11184	C_6H_6	5.349	412.3	Y	TOLUENE
BUTANE	58.122200	C_4H_{10}	4.687	531.4	Y	PROPANE
CARBON	12.0107	C	2.94	74.8		
CARBON DIOXIDE	44.009500	CO_2	3.941	195.2		CARBON DIOXIDE
CARBON MONOXIDE	28.010100	CO	3.690	91.7	Y	CARBON MONOXIDE
CHLORINE	70.906	Cl_2	4.217	316.0	Y	
DODECANE	170.33484	$C_{12}H_{26}$	4.701	205.78	Y	N-HEPTANE
ETHANE	30.069040	C_2H_6	4.443	215.7	Y	ETHANE
ETHANOL	46.068440	C_2H_5OH	4.530	362.6	Y	METHANOL
ETHYLENE	28.053160	C_2H_4	4.163	224.7	Y	ETHYLENE

WRITING AN FDS INPUT FILE: &REAC

Example 1

```
&REAC          FUEL = 'N-HEPTANE' /
```

n-Heptane is in the library. It is assumed that the soot and CO yields are zero. FDS computes the yields of product species and the heat of combustion based upon predefined values

Example 2

```
&REAC          FUEL = 'N-HEPTANE'
        SOOT_YIELD = 0.02
          CO_YIELD = 0.10
HEAT_OF_COMBUSTION = 44600. /
```

The values of soot yield, CO yield and heat of combustion are overwritten with respect to the predefined ones relative to n-Heptane

FIRE DYNAMICS SIMULATOR

WRITING AN FDS INPUT FILE: &REAC

Example 3

```
&REAC          FUEL = 'NEWFUEL'
           FORMULA = 'C4H8O3N4'
HEAT_OF_COMBUSTION = 42000. /
```

or

```
&REAC          FUEL = 'NEWFUEL'
                 C = 4 , H = 8, O = 3, N = 4
HEAT_OF_COMBUSTION = 42000. /
```

If the fuel is not in the library, FDS uses the gas thermophysical properties of ETHYLENE along with the molecular weight given by the FORMULA or the values of C, H, O and N.

WRITING AN FDS INPUT FILE: &OBST

It defines rectangular solids inside the computational domain

```
&OBST XB= 3.5, 4.5,-1.0, 1.0, 0.0, 0.0, SURF_ID='STEEL FLANGE'
```

XB defines the volume with a sextuplet.
SURF_ID it applies a boundary conditions to an obstruction surface which is defined by the SURF namelist.

WRITING AN FDS INPUT FILE: &SURF

It defines the structure of all solid surfaces or openings within or bounding the flow domain. Boundary conditions for obstructions and vents are prescribed by referencing the appropriate SURF line(s).

Example: square fire

```
&SURF ID = 'POOL_FIRE’, HRRPUA = 1000.0 /

&OBST XB= -0.5,0.5,-0.5,0.5,0.0,0.1, SURF_IDS = 'POOL_FIRE', 'INERT',
'INERT' /
```

HRRPUA Heat Release Rate Per Unit Area (kW/m^2)
SURF_IDS an array of three character strings specifying the boundary condition IDs for the top, sides and bottom of the obstruction.
INERT solid surface with temperature fixed at TMPA (default)

WRITING AN FDS INPUT FILE: &SURF

Example: square fire

```
&SURF ID = 'POOL_FIRE', HRRPUA = 1.0, RAMP_Q='FIRE' /

&RAMP ID='FIRE', T=0.0,   F=0.0 /
&RAMP ID='FIRE', T=1.0,   F=1000. /
```

RAMP_Q Time history of HRR as a ramp

WRITING AN FDS INPUT FILE: &SURF

Useful parameters

SURF (Surface Properties)				
ADIABATIC	Logical	Adiabatic thermal BC		.FALSE.
EMISSIVITY	Real	Emissivity		0.9
HRRPUA	Real	HRR Per Unit Area	kW/m^2	0.
ID	Character	IDentifier		
RAMP_Q	Character	Ramp ID for HRR		
RGB(3)	Int. Triplet	Color indices (0-255)		255,204,102
TMP_FRONT	Real	Front surface temperature	°C	20.
VEL	Real	Normal velocity	m/s	0.

WRITING AN FDS INPUT FILE: &VENT

It is used to prescribe planes adjacent to obstructions or external walls. A `VENT` must always be attached to a solid obstruction. It can be used:

- For ducts
- Defining fire on an `OBST`

```
&OBST XB=0.0,5.0,2.0,3.0,0.0,4.0, SURF_ID='big block' /
&VENT XB=1.0,2.0,2.0,2.0,1.0,3.0, SURF_ID='hot patch' /
```

- Defining an open wall

```
&VENT MB='XMIN', SURF_ID='OPEN' /
```

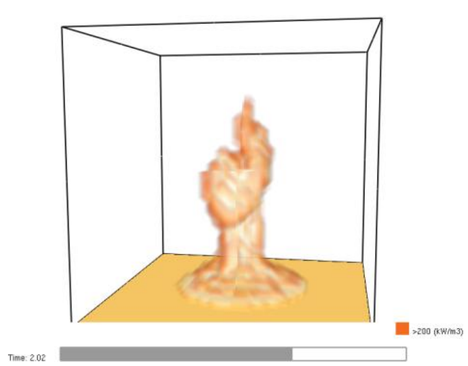
`MB` specifies an entire open external wall of plane with x = `XMIN`

NOTE: By default, it is assumed that ambient conditions exist beyond the `OPEN` vent. Sometimes it is good to enlarge the domain beyond the compartment to well reproduce the flow pattern from a door or window.

WRITING AN FDS INPUT FILE: &VENT

- Circular vents

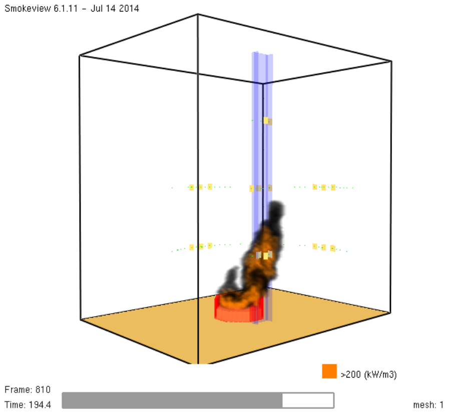
```
&SURF ID = 'POOL_FIRE’, HRRPUA = 620. /
&VENT XB=-0.5,0.5,-0.5,0.5,0,0, XYZ=0,0,0, RADIUS=0.5,
SURF_ID=‘POOL_FIRE'/
```



WRITING AN FDS INPUT FILE: &VENT

- Wind effects at OPEN boundaries imposed according to a pressure (Pa)

```
&VENT MB='YMAX', SURF_ID='OPEN', DYNAMIC_PRESSURE = 0.25 /
```

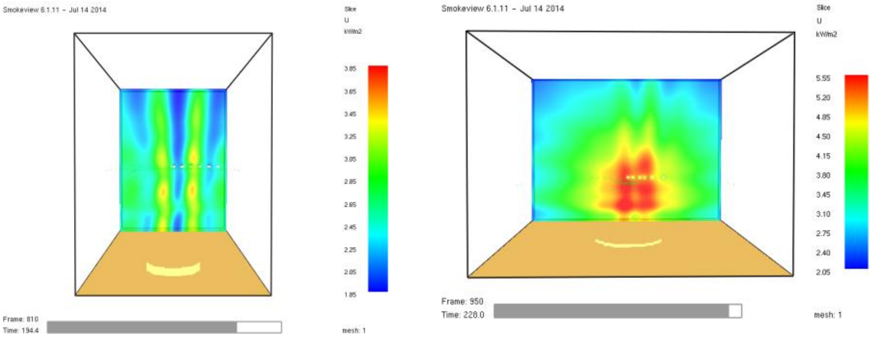


FIRE DYNAMICS SIMULATOR

WRITING AN FDS INPUT FILE: &RADI

It contains all of the parameters related to the radiation solver

```
&RADI NUMBER_RADIATION_ANGLES=200
      RADIATIVE_FRACTION=0.40 /
```



The number of radiation angles should be increased if the measurement point is located fairly away from the fire source.

WRITING AN FDS INPUT FILE: &DEVC (CHID_devc.csv)

It defines the location and the quantity to read by the devices

```
&DEVC ID='TH1', XYZ=3.0,5.6,2.3, QUANTITY='TEMPERATURE'/
```

You will read in the `CHID_devc.csv` the evolution of the gas temperature at that location with the string `TH1` at the top of the column.

```
&DEVC ID='HF1', XYZ=0.25,0.5,0.5, QUANTITY='RADIATIVE HEAT FLUX GAS',
ORIENTATION=1,0,0  /
```

`RADIATIVE HEAT FLUX GAS` records the radiative heat flux away from a solid surface

$$\dot{q}'' = \left(\dot{q}''_{inc.rad} - \sigma T_{MPA}{}^{4}\right)$$

$\dot{q}''_{inc.rad}$ incident radiation

σ Stefan-Boltzmann constant

T_{MPA} ambient temperature

WRITING AN FDS INPUT FILE: &SLCF (CHID_n.sf)

Animated planar slices

```
&SLCF PBX=0.2, QUANTITY='TEMPERATURE'/
&SLCF PBX=0.2, QUANTITY='VELOCITY'/
&SLCF PBX=0.2, QUANTITY='INTEGRATED INTENSITY'/
```

`PBX` The quantity will be read in the plane for which x = 0.2 m

`INTEGRATED INTENSITY` It is useful to check if the number of radiation angles is sufficient to measure the heat flux at the specified location

NOTE: FDS averages slice file data at cell corners. For example, gas temperatures are computed at cell centers, but they are linearly interpolated to cell corners and output to a file that is read by Smokeview. To prevent this from happening, set `CELL_CENTERED=.TRUE.`

FIRE DYNAMICS SIMULATOR

WRITING AN FDS INPUT FILE: **`&BNDF (CHID_n.bf)`**

Animated boundary quantities: record surface quantities at all solid obstructions

```
&BNDF QUANTITY = ‘RADIATIVE HEAT FLUX' /
&BNDF QUANTITY = ‘WALL TEMPERATURE' /
&BNDF QUANTITY = ‘INCIDENT HEAT FLUX' /
```

FIRE DYNAMICS SIMULATOR

WRITING AN FDS INPUT FILE: **`&ISOF (CHID_n.iso)`**

Animated isosurfaces: creates three-dimensional animated contours of gas phase scalar quantities

```
&ISOF QUANTITY='TEMPERATURE', VALUE(1)=50., VALUE(2)=200.,
VALUE(3)=500. /

&ISOF QUANTITY=‘U-VELOCITY', VALUE(1)=0.1, VALUE(2)=1., VALUE(3)=5. /
```

FIRE DYNAMICS SIMULATOR

SIMULATION OUTPUT: CHID.out

```
Run Time Diagnostics

      Time Step       1   April 19, 2017  12:03:01
      Step Size:    0.100E+00 s, Total Time:       0.10 s
      Pressure Iterations:      1
      Maximum Velocity Error:  0.00E+00 on Mesh   1 at (   0   0   0)
      Maximum Pressure Error:  0.25E+01 on Mesh   1 at (  10  10   1)
      ---------------------------------------------------------------
      Max CFL number:  0.55E-01 at (  10,   1,   1)
      Max divergence:  0.34E+00 at (  10,  10,  10)
      Min divergence: -0.86E-01 at (   5,   6,   6)
      Max VN number:   0.14E-01 at (   2,   7,   4)
      No. of Lagrangian Particles:           2
      Radiation Loss to Boundaries:        4.410 kW

      Time Step       2   April 19, 2017  12:03:01
      Step Size:    0.100E+00 s, Total Time:       0.20 s
      Pressure Iterations:      1
      Maximum Velocity Error:  0.00E+00 on Mesh   1 at (   0   0   0)
      Maximum Pressure Error:  0.10E+02 on Mesh   1 at (   1  10   1)
      ---------------------------------------------------------------
      Max CFL number:  0.58E-01 at (  10,   1,   1)
      Max divergence:  0.32E+00 at (   1,   1,  10)
      Min divergence: -0.83E-01 at (   5,   6,   6)
      Max VN number:   0.15E-01 at (   2,   7,   4)
      No. of Lagrangian Particles:           2
      Radiation Loss to Boundaries:        4.404 kW
```

FIRE DYNAMICS SIMULATOR

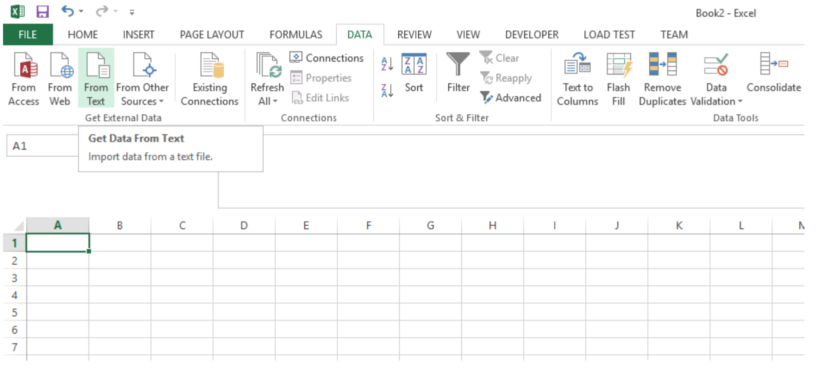
SMOKEVIEW: CHID.smv

- You can visualize the simulation results with SMOKEVIEW
- The CHID.smv is an ASCII file so you can see its content with a text editor
- Look at Appendix C of the Smokeview manual to see useful keyboard shortcuts
    - *t*: start/stop animation
    - *o*: show/hide domain boundaries
    - *u*: reload the current results
    - !: snap scenes's view angle
    - alt+*v*: switch between perspective and size preserving view
    - *g*: show/hide grid
    - *m*: change mesh
    - *r*: produce a picture file of the scene

SMOKEVIEW: DATA ELABORATION

Data can be very large and plan carefully what to save before running the analysis because it cannot be done after it.

For files *.csv use EXCEL



FIRE DYNAMICS SIMULATOR

PRACTICAL SESSION

Exercise 1: Open compartment with localised fire

- **1-1** Mesh
- **1-2** Obstructions and Vents
- **1-3** Fire
- **1-4** Devices
- **1-5** Change parameters

Exercise 2: Radiation box

Exercise 3: Pool fire with assigned HRR

Exercise 4: Pool fire mesh refinement and engulfed steel column

Exercise 5: Pool fire with flame touching a ceiling

PRACTICAL SESSION: Exercise 1-1

- **Compartment - MESH**
  - Define:
    - **Domain dimensions**: 7 m x 7 m x 3.5 m
    - **Mesh:** 10 cm x 10 cm x 10 cm
    - **Turbulence model:** Constant Smagorinsky
    - **Analysis time:** 10 s
    - **Number of radiation angles:** 200

MESH CONSIDERATIONS

- Try to use cell shapes as cubic as possible
- Be aware that in an LES formulation you need to well-resolve the large eddies to have good results
- The **mesh size** should be based on sensitivity grid analysis and depends on the HRR through the characteristic fire diameter

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

$\dot{Q}$ total release rate (kW)

ρ_∞ air density (kg/m^3)

c_p specific heat of air (kJ/kgK)

T_∞ air temperature (K)

g gravity acceleration (m/s^2)

MESH CONSIDERATIONS

Different studies suggest a minimum ratio between D* anf δx for the estimate of plume temperatures and plume heights.

If HRR changes with time also the optimal mesh size varies

-Ma and Quintiere / Fire Safety Journal 38 (2003) 467-492)

-FDS Validation Guide

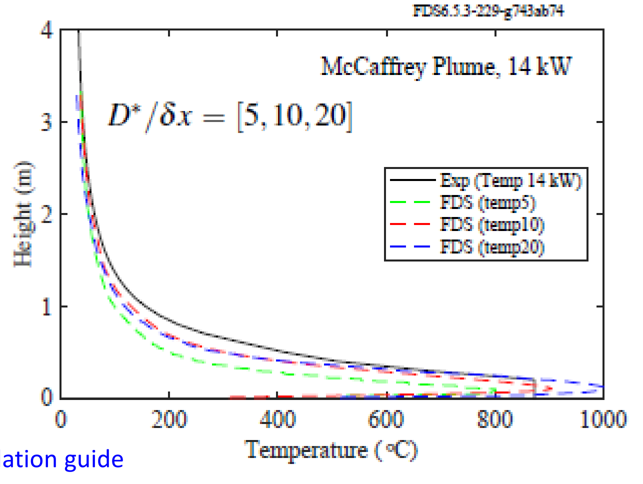
$$D^*/\delta x > 10 \div 20$$

$$\delta x \quad \text{max cell size (m)}$$

FIRE DYNAMICS SIMULATOR

MESH CONSIDERATIONS

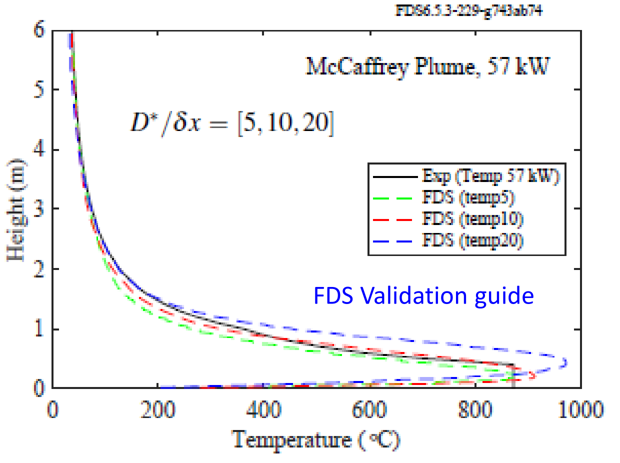
Plume temperature



FDS Validation guide

MESH CONSIDERATIONS

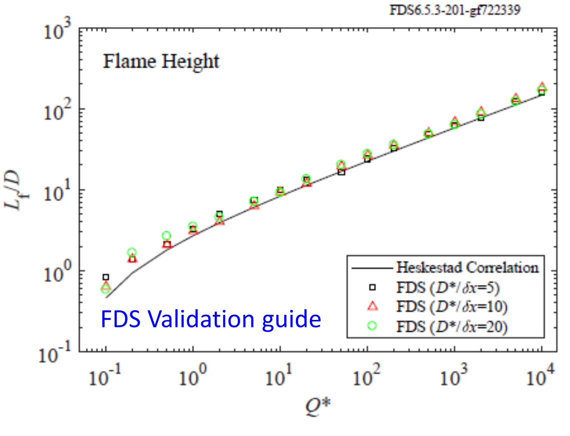
Plume temperature



FIRE DYNAMICS SIMULATOR

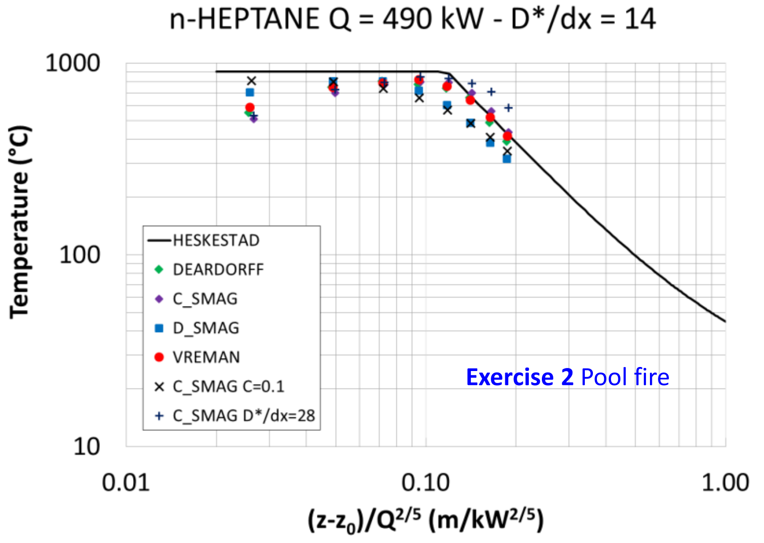
MESH CONSIDERATIONS

Flame height (FDS flame height prediction based on the fuel consumption, e.g. 99% or 95%)



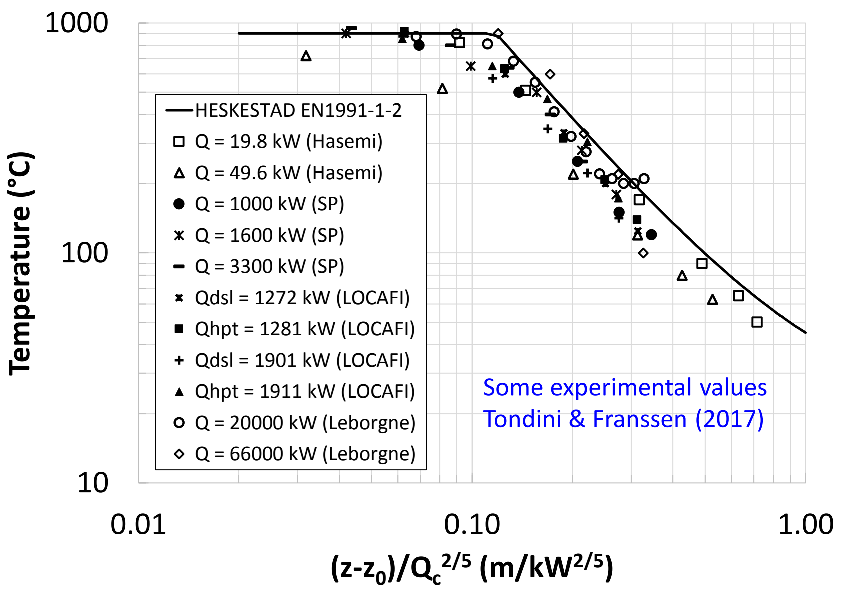
FIRE DYNAMICS SIMULATOR

MESH CONSIDERATIONS



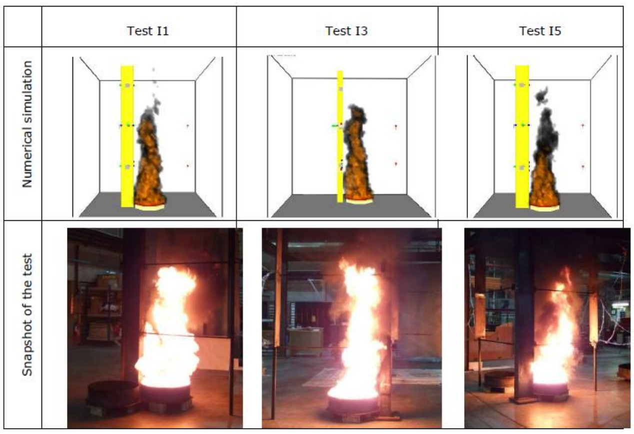
FIRE DYNAMICS SIMULATOR

MESH CONSIDERATIONS



MESH CONSIDERATIONS

In the RFCS LOCAFI project the turbulence model that could be represent the experimental results was the Constant Smagorinsky model with C_s=0.1

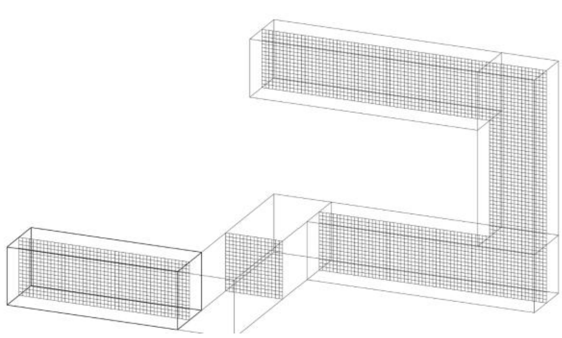


FIRE DYNAMICS SIMULATOR

MESH CONSIDERATIONS

Multiple meshes

The computational domain consists of more than one computational mesh, usually connected although this is not required. If more than one mesh is used, there should be a `MESH` line for each**.**



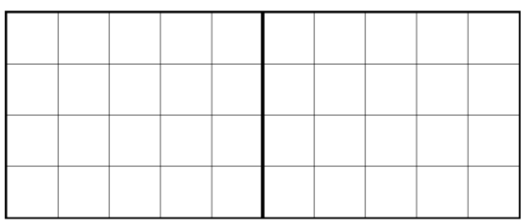
FIRE DYNAMICS SIMULATOR

MESH CONSIDERATIONS

Multiple meshes

If an MPI Parallel computing is used the mesh must be divided into several meshes to be run on different computers

Mesh alignment

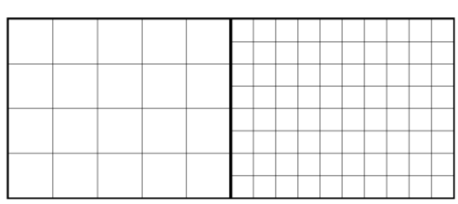


This is the ideal kind of mesh to mesh alignment.

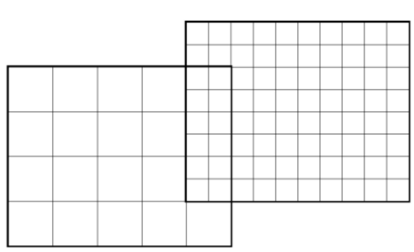
FIRE DYNAMICS SIMULATOR

MESH CONSIDERATIONS

Mesh alignment



This is allowed so long as there are an integral number of fine cells abutting each coarse cell.

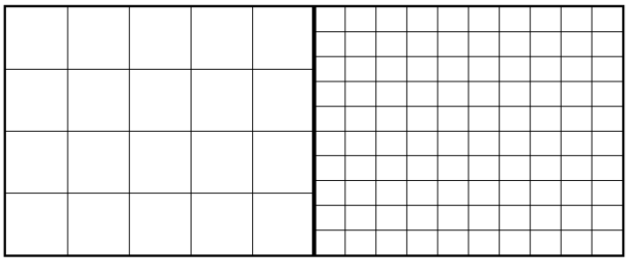


This is allowed, but of questionable value.

The information is only exchanged at the exterior boundaries

MESH CONSIDERATIONS

Mesh alignment



This is not allowed.

THERMAL PROPERTIES OF SOLID OBJECTS

It is important to define the thermal properties of the object that are in the computational domain because they influence the fire development.

- The default condition of each solid boundary is `INERT`, i.e. fixed ambient temperature.
- `SURF` line defines the heat transfer characteristics of the obstruction, i.e. how the obstruction is seen by the 1D solid phase solver
- Solids are assumed to consist of layers that can be made of different materials.
- The properties of each material required are designated via `MATL`
- **FDS has a 1D solid phase solver. Not always enough for structural elements**

THERMAL PROPERTIES OF SOLID OBJECTS

Example

```
&MATL  ID='BRICK', CONDUCTIVITY=0.69, SPECIFIC_HEAT=0.84,
       DENSITY=1600., EMISSIVITY = 0.85 /
&SURF  ID='BRICK WALL', MATL_ID='BRICK', COLOR='RED',
       BACKING='EXPOSED', THICKNESS=0.20 /
&OBST  XB=0.1,5.0,1.0,1.2,0.0,1.0, SURF_ID='BRICK WALL' /
```

Note that the thickness of the wall indicated by the OBST line is independent of the THICKNESS specified by the SURF line.

The 1D solid phase solver considers THICKNESS

FIRE DYNAMICS SIMULATOR

THERMAL PROPERTIES OF SOLID OBJECTS

Thermal properties can vary with temperature

```
&MATL ID='STEEL_EN1993-1-2'
      EMISSIVITY = 0.7
      CONDUCTIVITY_RAMP = 'k'
      SPECIFIC_HEAT_RAMP = 'cp'
      DENSITY = 7850.0 /

&RAMP ID='cp',      T =    20      ,F =  0.440  /
&RAMP ID='cp',      T =    60      ,F =  0.466  /
&RAMP ID='cp',      T =    100     ,F =  0.488  /
&RAMP ID='cp',      T =    140     ,F =  0.506  /
```

SOLID SURFACE TEMPERATURE

The surface temperature can be fixed

```
&SURF ID='HOT WALL', COLOR='RED', TMP_FRONT=200. /
```

There is no need to specify a `MATL_ID` or `THICKNESS` because the wall is to be maintained at the given temperature.

CONVECTIVE HEAT TRANSFER

In an LES calculation, the convective heat transfer coefficient is based on a combination of natural and forced convection correlations

$$h = \max\left[C\left|T_g - T_m\right|^{1/3}, \frac{k}{L}\mathrm{Nu} \right]$$

$$\dot{q}'' = h\left(T_g - T_m\right)$$

C empirical coefficient for natural convection

L characteristic length related to the size of the physical obstruction

k thermal conductivity of the gas

Nu Nusselt number

CONVECTIVE HEAT TRANSFER

The convective heat transfer coefficient can be specified explicitly using `HEAT_TRANSFER_COEFFICIENT` in `SURF`

ADIABATIC SURFACES

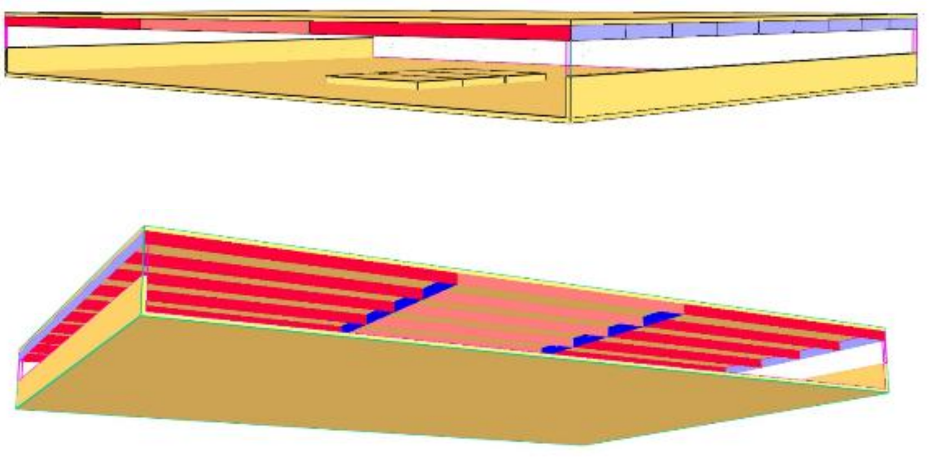
A surface can be defined as adiabatic, i.e. zero neat heat flux from the gas to the solid.

`ADIABATIC=.TRUE.` in `SURF` which means that the `NET_HEAT_FLUX=0` and `EMISSIVITY=1`.

NOTE: no real surface is perfectly adiabatic.

ADIABATIC SURFACES

Example: Open car park with beam defined as adiabatic surfaces



HEAT CONDUCTION IN SOLIDS

`MATL` is used to define the properties of the materials that make up boundary solid surfaces.

A solid boundary can consist of **multiple layers of different materials**, and **each layer can consist of multiple materials**.

`MATL(IL,IC)`

The argument `IL` is an integer indicating the layer index, starting at 1, the layer at the exterior boundary. The argument `IC` is an integer indicating the component index.

FIRE DYNAMICS SIMULATOR

HEAT CONDUCTION IN SOLIDS

Example 2 layers of 2 materials. 1) MAT1 2) INSULATOR

```
&MATL ID = ‘MAT1‘, CONDUCTIVITY = 0.1, SPECIFIC_HEAT = 1.0
DENSITY = 500. /
&MATL ID = 'INSULATOR‘, CONDUCTIVITY = 0.041, SPECIFIC_HEAT = 2.09
DENSITY = 229. /
&SURF ID = 'BRICK WALL‘, MATL_ID = ‘MAT1','INSULATOR‘
BACKING = 'EXPOSED‘, THICKNESS = 0.20,0.10 /
```

FIRE DYNAMICS SIMULATOR

HEAT CONDUCTION IN SOLIDS

If BACKING =‘EXPOSED’, the last MATL_ID is applied to the opposite face of the OBST, assuming that the OBST is zero or one grid cells thick.

If the obstruction is less than or equal to one cell thick, then the innermost layer will be exposed to the air temperature on the back side.

If the obstruction is on the boundary of the domain or is more than one cell thick, it is assumed to back up to an air gap at ambient temperature.

⇩

Heat can be transferred through the wall

FIRE DYNAMICS SIMULATOR

HEAT CONDUCTION IN SOLIDS

A layer can be made of multiple materials

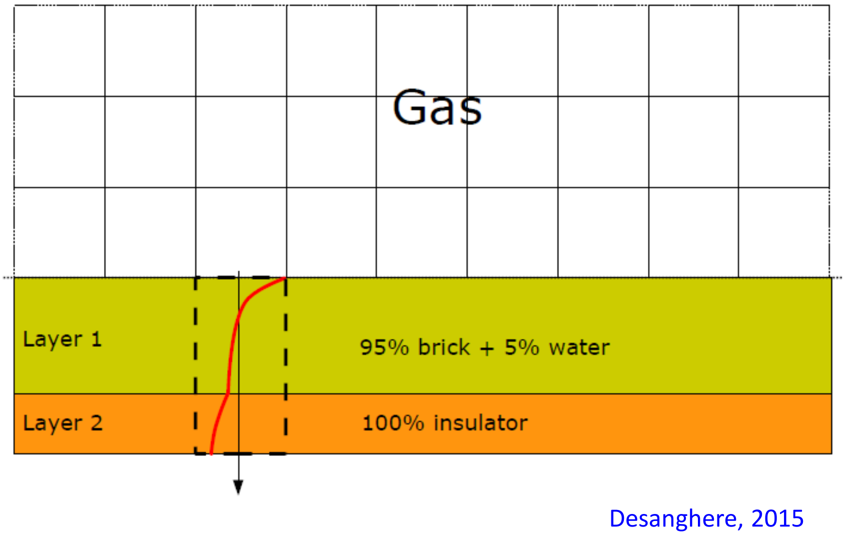
```
&MATL ID = 'WATER‘, CONDUCTIVITY = 0.60, SPECIFIC_HEAT = 4.19
DENSITY = 1000. /
&SURF ID = 'BRICK WALL‘
MATL_ID(1,1:2) = 'BRICK','WATER‘
MATL_MASS_FRACTION(1,1:2) = 0.95,0.05
MATL_ID(2,1) = 'INSULATOR'
     BACKING = 'EXPOSED'
   THICKNESS = 0.20,0.10 /
```

Material 1 = 95% brick & 5% water

Material 2

FIRE DYNAMICS SIMULATOR

HEAT CONDUCTION IN SOLIDS



Desanghere, 2015

PRACTICAL SESSION: Exercise 1-2

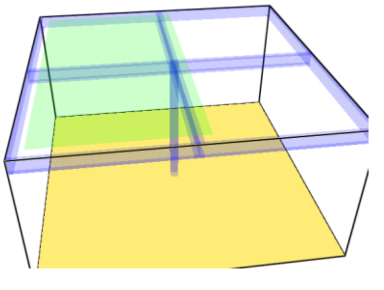
- **Compartment – OBST & VENT**
  - Define the materials:
    - Steel – Thermal properties according to EN1993-1-2
    - Plaster `EMISSIVITY = 0.9, CONDUCTIVITY = 0.2, SPECIFIC_HEAT = 1.7, DENSITY = 800.`
  - Define `OPEN` boundary conditions at each side of the compartment
  - The ceiling, for example, covers only half of the compartment

PRACTICAL SESSION: Exercise 1-2

- **Compartment – OBST & VENT**
  - Define the obstructions forming the steel profiles

Profiles SHS 200 mm x 10 mm & IPE 300
Ceiling Plaster board – t = 0.01 m

  - SHS is a column located at the middle of the compartment
  - The IPE profiles are perimetric beams



FIRE MODELS

- Fire with a specified Heat Release Rate (Recommended)

```
&SURF ID = 'POOL_FIRE’, HRRPUA = 500. /
```

- Radially spreading fire

```
&VENT    XB=-0.6,0.6,-0.6,0.6,0,0,    XYZ=0,0,0,    SPREAD_RATE=0.05,
COLOR='BLUE', SURF_ID='BURNER' /
```

`XYZ` point from which the fire starts

`SPREAD_RATE` spread rate in m/s

- Pyrolysis models (Not recommended)

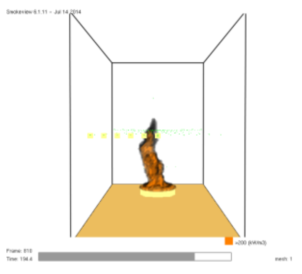
PRACTICAL SESSION: Exercise 1-3

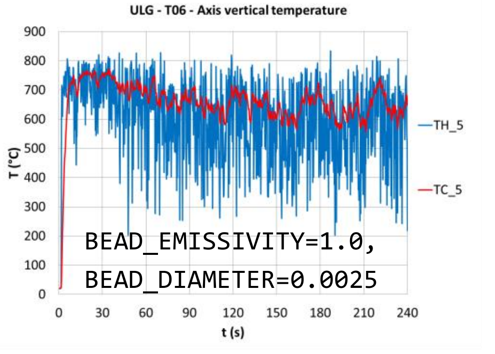
- **Compartment - FIRE**
  - Define the fire source:
    - **Dimension:** 1 m x 1 m
    - **Position:** `XB=3.0,4.0,1.0,2.0,0.0,0.16`
    - **HRR:** 3 MW
    - **REAC:** $C_{12}H_{24}$

OUTPUT QUANTITIES (&DEVC)

`TEMPERATURE` records the temperature of the gas

`THERMOCOUPLE` is the temperature of a modeled thermocouple. The thermocouple temperature lags the true gas temperature by an amount determined mainly by its bead size.





FIRE DYNAMICS SIMULATOR

OUTPUT QUANTITIES (&DEVC)

- `NET HEAT FLUX` records the net heat flux

$$\dot{q}''_{net} = \varepsilon_s \left(\dot{q}''_{inc.rad} - \sigma T_s^4 \right) + h \left(T_g - T_s \right)$$

$\dot{q}''_{inc.rad}$ incident radiative heat flux

ε_s surface emissivity

- `CONVECTIVE HEAT FLUX` records the net convective heat flux

$$\dot{q}''_{conv} = h \left(T_g - T_s \right)$$

- `RADIATIVE HEAT FLUX` records the net radiative heat flux

$$\dot{q}''_{rad} = \varepsilon_s \left(\dot{q}''_{inc.rad} - \sigma T_s^4 \right)$$

OUTPUT QUANTITIES (&DEVC)

- `GAUGE HEAT FLUX` predicts the heat flux obtained from a measurement

$$\dot{q}''_{gauge} = \varepsilon_{gauge}\left(\dot{q}''_{inc.rad} - \sigma T^4_{gauge}\right) + h\left(T_g - T_{gauge}\right)$$

$\dot{q}''_{inc.rad}$ incident radiative heat flux

ε_{gauge} gauge emissivity (default 1.0)

T_{gauge} gauge temperature (TMPA)

- `RADIOMETER` measures only the radiative heat flux

$$\dot{q}''_{radmtr} = \varepsilon_{gauge}\left(\dot{q}''_{inc.rad} - \sigma T^4_{gauge}\right)$$

NOTE: all previous flux devices must attached to a surface

- `RADIATIVE HEAT FLUX GAS` records the rad heat flux away from a surface

$$\dot{q}''_{flxgas} = \left(\dot{q}''_{inc.rad} - \sigma T_{MPA}{}^4\right)$$

FIRE DYNAMICS SIMULATOR

OUTPUT QUANTITIES (&DEVC)

- `WALL TEMPERATURE` is a quantity that measures the temperature of the a solid. The device must be attached to the solid.

```
&DEVC ID = 'TW1‘, QUANTITY = 'WALL TEMPERATURE’, XYZ = 3.5,2.4,1.0,
IOR = -2 /
```

- `ADIABATIC SURFACE TEMPERATURE` is a quantity representative of the heat flux to a solid surface

$$\varepsilon_s\left(\dot{q}''_{inc.rad} - \sigma T^4_{AST}\right) + h\left(T_g - T_{AST}\right) = 0$$

T_{AST} temperature for which the net heat flux is zero

It provides the gas phase thermal boundary condition in a single quantity.

OUTPUT QUANTITIES (&DEVC)

You can output the **adiabatic surface temperature even when there is no actual solid** surface and provides the maximum achievable solid surface temperature at the given location XYZ that is not actually in the vicinity of a solid surface.

```
&DEVC ID='AST', XYZ=...,QUANTITY='ADIABATIC SURFACE TEMPERATURE GAS',
ORIENTATION=1.0,0.0,0.0, PROP_ID='props' /
&PROP ID='props', EMISSIVITY=0.7, HEAT_TRANSFER_COEFFICIENT=9. /
```


PRACTICAL SESSION: Exercise 1-4

- **Compartment - DEVC**
  - Insert devices into the model at different locations

```
'TEMPERATURE'
'THERMOCOUPLE'
'WALL TEMPERATURE'
'ADIABATIC SURFACE TEMPERATURE'
'ADIABATIC SURFACE TEMPERATURE GAS'
```

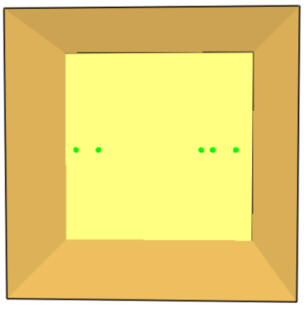
  - Insert animated planar slices

FIRE DYNAMICS SIMULATOR

PRACTICAL SESSION: Exercise 2

- **Radiation box** with all walls at fixed temperature of 500°C
  - Check that there is no symmetry when reading the temperature
  - Change the temperature of one or more walls and see how the flow establishes in the domain

Dimensions 1 m x 1 m x 1 m
Mesh size 10 cm x 10 cm x 10 cm
Wall emissivity 1.0

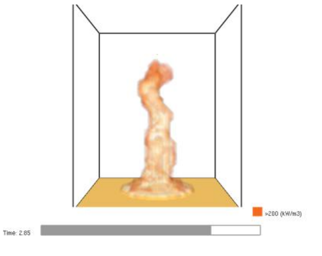


FIRE DYNAMICS SIMULATOR

PRACTICAL SESSION: Exercise 3

- **Pool fire** with assigned HRR
  - Change the turbulence model and see how the flame characteristics vary
  - Record the temperature along the axis and compare them with the Heskestad model of EN1991-1-2
  - Change the number of cores and check the computational time

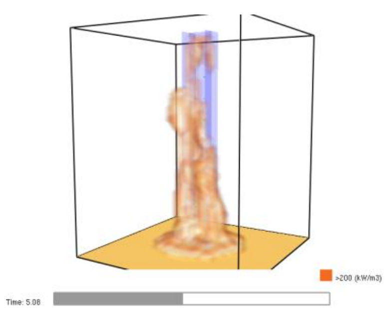
HRR 620 kW/m^2
Fuel n-Heptane
Diameter 1.0 m
Domain dimensions 2 m x 2 m x 2.5 m
Mesh size 5 cm x 5 cm x 5 cm

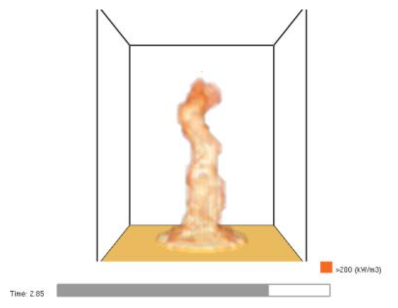


FIRE DYNAMICS SIMULATOR

PRACTICAL SESSION: Exercise 4

- **Consider the data of Exercise 2** Grid setting
  - Increase the mesh size and record the temperature along the axis and compare them with values taken in Exercise 2
  - Locate an HE 300 B profile at the centre of the fire and see the difference in flame height. Use material properties of EN1993-1-2



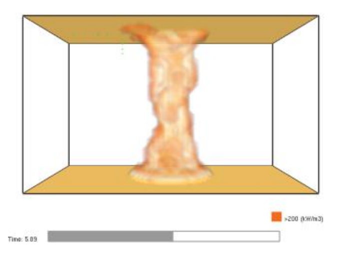


FIRE DYNAMICS SIMULATOR

PRACTICAL SESSION: Exercise 5

- **Pool fire with flame touching a ceiling**
  - Measure the heat flux with different devices at the ceiling level
  - Change the material of the ceiling: adiabatic, concrete t = 15 cm, steel sheeting t = 2 mm. Use material properties of EN Part 2
  - Compare the results with the Hasemi model of EN1991-1-2
  - Change the number of radiation angles

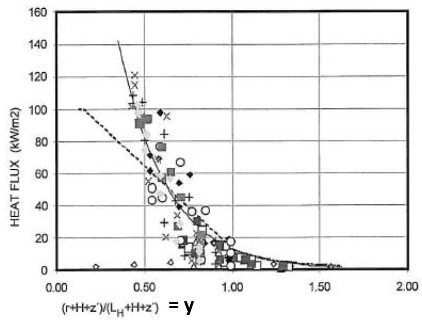
HRR 1800 kW/m^2
Fuel Propane
Diameter = 1.6 m
Domain dimensions 5 m x 3 m x 3 m
Mesh size 10 cm x 10 cm x 10 cm

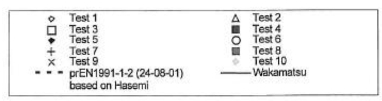


FIRE DYNAMICS SIMULATOR

PRACTICAL SESSION: Exercise 5

		HEAT FLUX (kW/m²)						
r (m)	y	HASEMI	ADBT C_SMAG	CONC C_SMAG	STEEL C_SMAG	ADBT VREMAN	CONC VREMAN	STEEL VREMAN
1.0	0.770	43.1	50.11	43.49	40.65	49.69	49.30	47.58
1.3	0.828	36.1	38.02	32.22	28.35	34.05	33.75	32.80
1.5	0.867	31.4	31.81	26.78	22.45	26.88	26.56	25.66
1.8	0.924	24.4	24.15	19.97	16.07	19.51	19.31	18.15
2.0	0.963	19.8	19.97	16.36	12.94	16.05	16.03	14.70



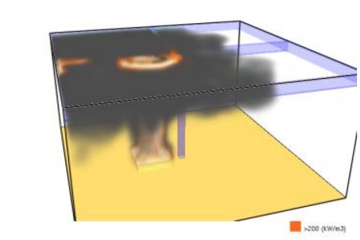


EN1991-1-2 Background document

FIRE DYNAMICS SIMULATOR

PRACTICAL SESSION: Exercise 1-5

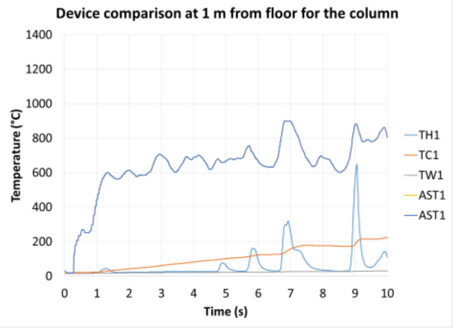
- **Compartment**
  - Measure the temperature with different devices near to the column and below the bottom flange of the beam.
  - Try to add wind in the compartment
  - See the smoke flow with and without beams

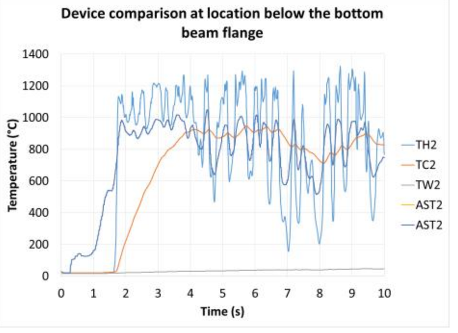


FIRE DYNAMICS SIMULATOR

PRACTICAL SESSION: Exercise 1

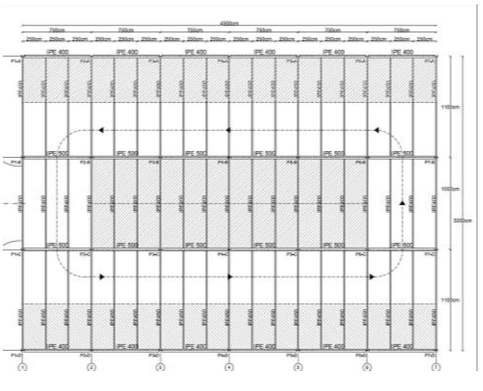
- **Compartment**

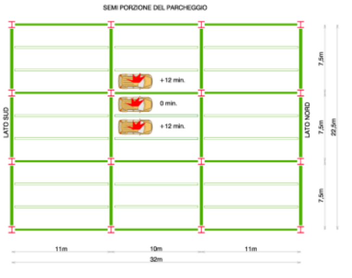


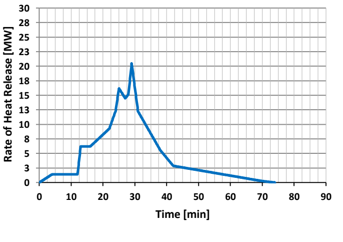


FIRE DYNAMICS SIMULATOR

EXAMPLE: Open steel-concrete composite car park







FIRE DYNAMICS SIMULATOR

EXAMPLE: Open steel-concrete composite car park

Initially **no structural elements** across the compartment were included in the CFD model.

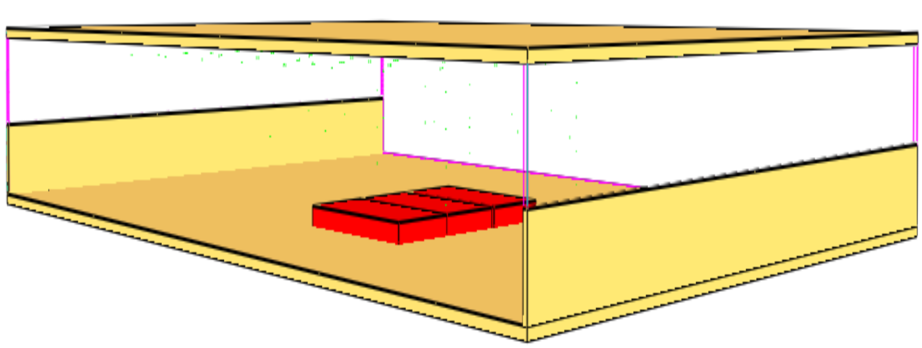
The **boundary conditions were consistently modelled** with physical parameters that described the thermal properties of the concrete slab, floor and parapets.

The burning cars were simulated by assigning to obstructions located at 30 cm (at the level of the tyres) from the floor **the class 3 Car RHR curve**

```
&REAC          FUEL = ‘GASOLINE‘
                  C = 8, H = 18
         SOOT_YIELD = 0.22
HEAT_OF_COMBUSTION = 44400. /
```

FIRE DYNAMICS SIMULATOR

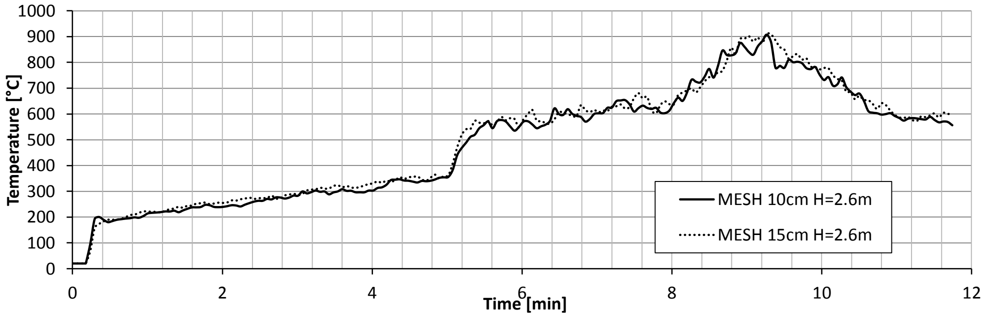
EXAMPLE: Open steel-concrete composite car park



FIRE DYNAMICS SIMULATOR

EXAMPLE: Open steel-concrete composite car park

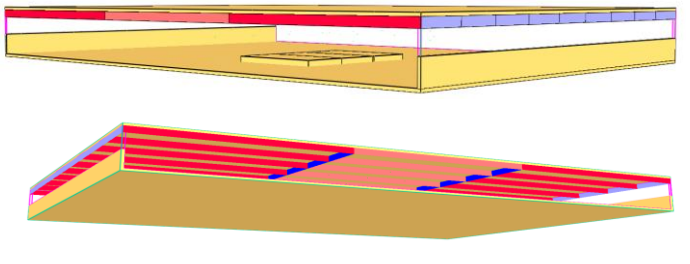
In order to provide realistic results the FDS mesh was selected by means of a sensitivity analysis.



FIRE DYNAMICS SIMULATOR

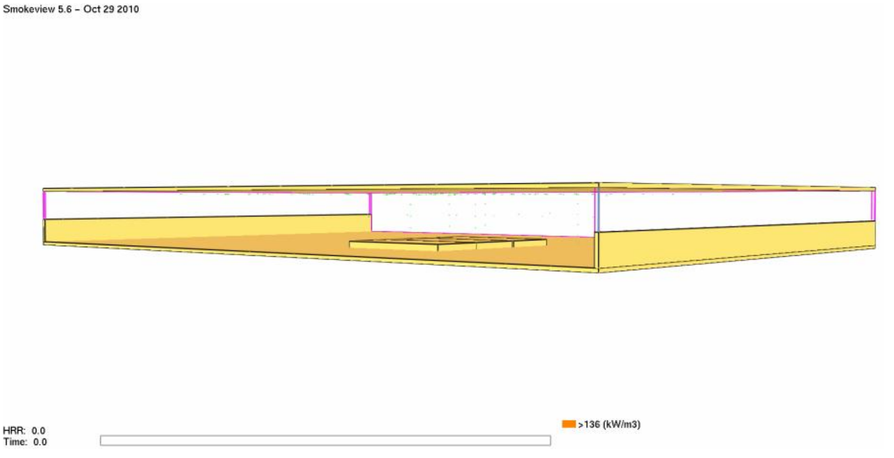
EXAMPLE: Open steel-concrete composite car park

The ratio between the beam and the ceiling heights suggests to include beams in the CFD model as they likely act as a barrier towards hot gases and smoke flow.



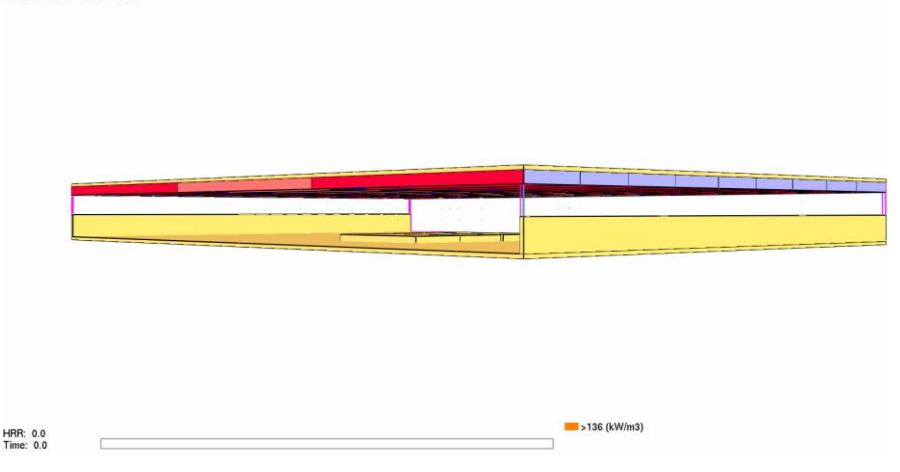
FIRE DYNAMICS SIMULATOR

EXAMPLE: Open steel-concrete composite car park



FIRE DYNAMICS SIMULATOR

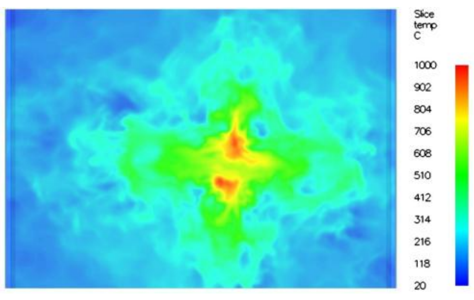
EXAMPLE: Open steel-concrete composite car park



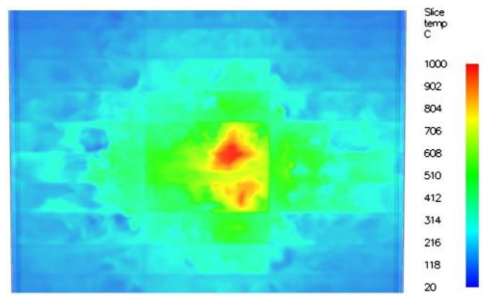
FIRE DYNAMICS SIMULATOR

EXAMPLE: Open steel-concrete composite car park

@ 30 min w/o beams



@ 30 min w beams

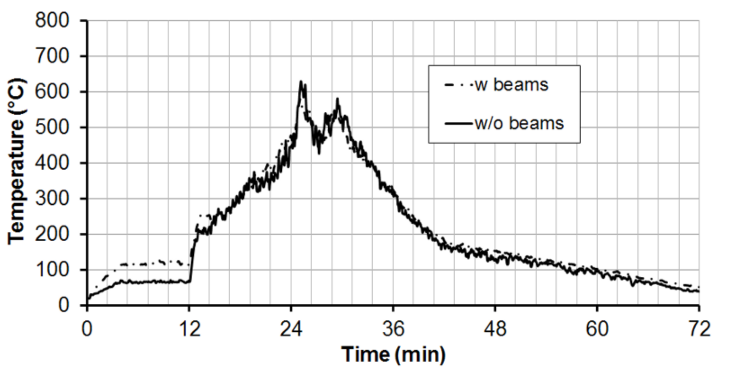


Tondini N., Morbioli A., Vassart O., Lechêne S., Franssen J.-M. (2016) An integrated modelling strategy between FDS and SAFIR: Methodology and application, Journal of Structural Fire Engineering, 7 (3), pp. 217-233.

FIRE DYNAMICS SIMULATOR

EXAMPLE: Open steel-concrete composite car park

Air temperature at the top of the column



N., Morbioli A., Vassart O., Lechêne S., Franssen J.-M. (2016) An integrated modelling strategy between FDS and SAFIR: Methodology and application, Journal of Structural Fire Engineering, 7 (3), pp. 217-233.

CONCLUSIONS

- FDS is a CFD software (with the limits of being free) to model the low-Mach fluid flow determined by fires.
- It is trickier to use with respect to nonlinear FE software. Convergence is most of the times achieved.
- Knowledge of different physical phenomena is needed.
- Check the FDS discussion group to solve issues and doubt. Read papers and books as much as possible.
- Start with simple models