



Software for Architecture,  
Engineering and Construction



# CYPEFIRE FDS

---

## USER MANUAL

User manual of the CYPEFIRE FDS application for the design of complex building models for the execution of fire evolution simulations using the FDS fluid dynamics computational standard ("Fire Dynamics Simulator") developed by NIST ("National Institute of Standards and Technology ", USA).





# Contents

<b>1 Basic Concepts.....</b>	<b>3</b>
1.1 Introduction .....	3
1.2 Initial Setup and Starting a Project .....	3
1.3 CYPEFIRE FDS interface .....	4
<b>2 BIM Model .....</b>	<b>8</b>
2.1 Libraries .....	8
2.2 Model .....	10
<b>3 FDS Model .....</b>	<b>11</b>
3.1 General data.....	11
3.2 Species .....	17
3.3 Materials.....	17
3.4 Surfaces .....	23
3.5 Particles .....	30
3.6 Reactions .....	37
3.7 Device models.....	41
3.8 Meshes.....	49
3.9 Zones.....	50
3.10 Obstructions.....	52
3.11 Holes .....	55
3.12 Vents.....	56
3.13 Dispositive .....	58
3.14 Controls .....	72
3.15 Sections.....	75
<b>4 Simulation .....</b>	<b>77</b>
4.1 Analysis/Calculation .....	77
4.2 Graphics.....	78

# 1 Basic Concepts

## 1.1 Introduction

Thank you for choosing CYPE and CYPEFIRE FDS. CYPEFIRE FDS is a program created for simulating the evolution of fires in complex building models using the computational standard of fluid dynamics FDS (Fire Dynamics Simulator) developed by NIST (National Institute of Standards and Technology, USA). This system is integrated in the Open BIM workflow via the BIMserver.center using the IFC standard. CYPEFIRE FDS allows users to import architectural data in the form of BIM models, define the elements of the model to enable them to be fed into the simulation engine, and then simulate the design conditions set by the user. The program carries out the necessary BIM model checks to ensure the model is defined correctly.

**Through the use of the SmokeViewer, the results of the analysis are able to be visualized and with tools included for controlling the simulation. The SmokeViewer includes tools for visualizing variables such as pressure, flow, temperature, etc.** The complexity involved in a dynamic simulation of fires in a building and the multiple factors involved mean that the hardware and software requirements of the machine where this simulation will be calculated are higher than what would be necessary to work normally with other CYPE programs.

## 1.2 Initial Setup and Starting a Project

This section of the manual will demonstrate how to start a project and begin using CYPEFIRE FDS. Begin by downloading the latest version of CYPEFIRE FDS from the BIMserver.center ([https://bimserver.center/en/store/cypefire\\_fds](https://bimserver.center/en/store/cypefire_fds)), you will require a BIMserver.center account, and an accessible IFC file model (Version 2019.e and above).

The first step is to open CYPEFIRE FDS. Next, connect to BIMserver.center on the far right side and sign in if needed by following the prompted steps. Once signed in, a new project can be created and an IFC file can be imported.

To continue working on a file, select **File Manager** under File and locate the desired project. Otherwise, to start a new project, select **New...** under File, choose where the file will be saved under Browse, and give the file a name and description if desired. After continuing, a screen will appear regarding the BIMserver.center. Here is where a new project within the BIMserver.center can be created by selecting "Create new project" or to

connect the project with an existing project select **Select project**. From the list select the appropriate project and click **Accept**.

The program will load the project selected and any files associated with the project. You can then select which files you would like to import for the simulation. The BIM model will automatically be imported as it is necessary for the simulation to function however; it is also possible to import a CYPFIRE Sprinklers file to enable the simulation to include sprinklers which are fully functional within the simulation. Clicking **Accept** will load the configuration selected and open the main interface of the program with your model loaded.

## 1.3 CYPEFIRE FDS interface

The CYPEFIRE FDS interface is comprised of two main areas separated by tabs at the top of the window. The two tabs are: *BIM Model*, and *FDS Model*.

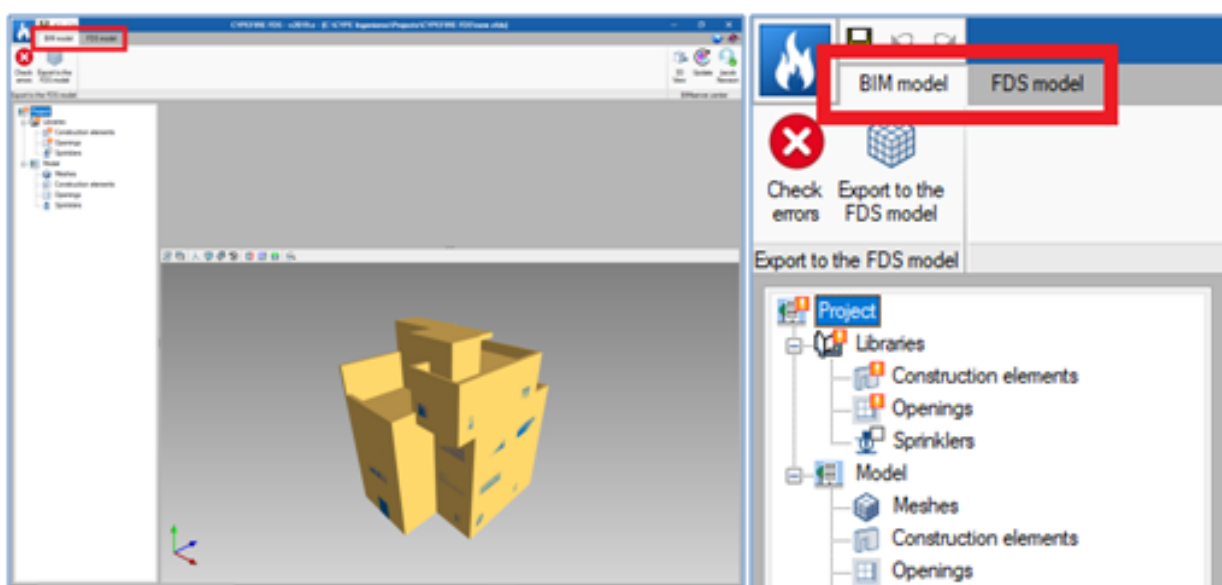


Fig. 1. FDS Interface

The BIM model tab is used to configure the necessary options to export the information of the BIM model into the FDS model. The FDS model tab is used for configuring the simulation itself. Both tabs are broken into 4 main sections: Primary toolbar (1), Selection panel (2), Project tree (3), and the 3D Model window (4). Each section is highlighted in the images below.

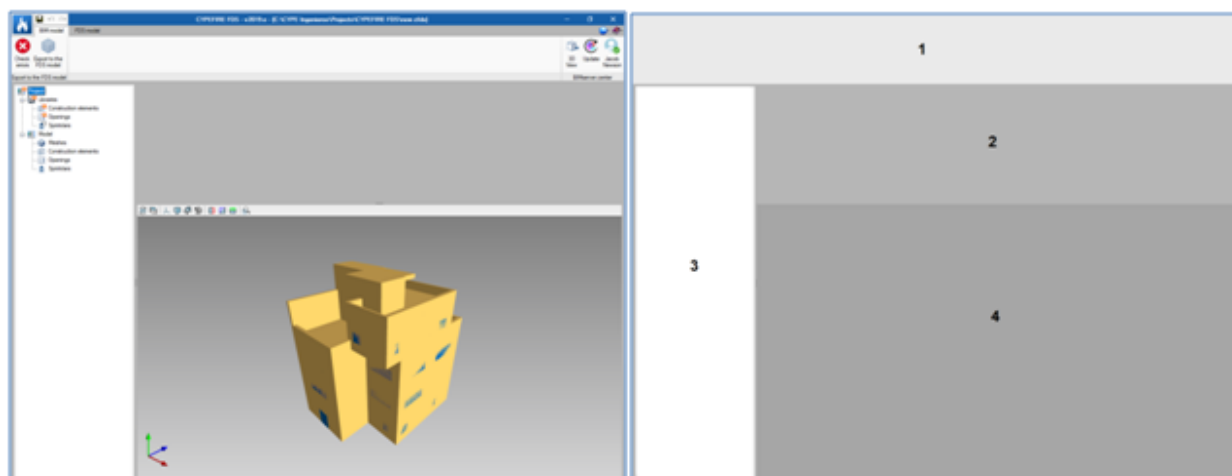


Fig. 2. Interface sections

### 1.3.1 *Primary toolbar*

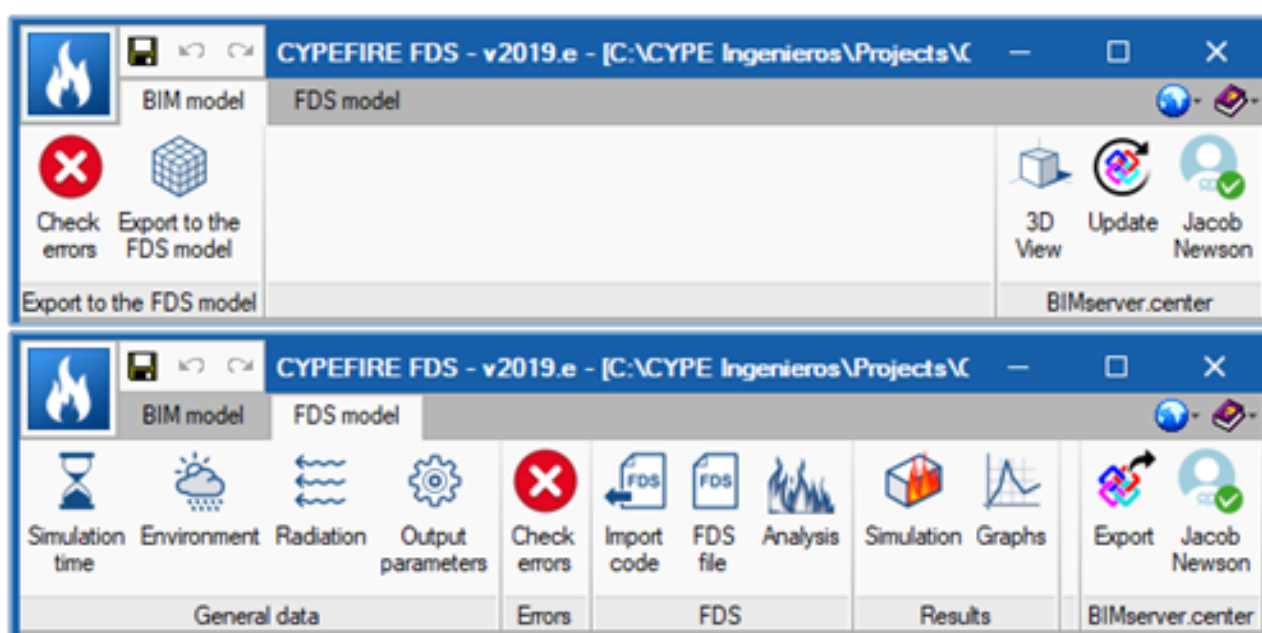


Fig. 3. Primary toolbars, "BIM Model" and "FDS Model" tabs

The primary toolbar contains the tools for controlling the configuration of the simulation. In the BIM model tab, this consists of checking the model for errors and exporting the model to the FDS model. The FDS model tab contains tools for altering the conditions of the simulation, conducting the analysis, accessing the simulation, etc.

### 1.3.2 *Selection panel*

Reference	Library	
WL1	Screed	
WL2	Floor slab	
WL3	Floor slab	
WL4	External floor slab	
WL5	Floor slab	
WL6	Floor slab	
WL7	Concrete roof 19	
WL8	Concrete roof 19	

Fig. 4. Selection panel

The selection panel is used to alter/input data for the corresponding element selected in the Project tree. It is possible to edit elements in the list, add new elements, delete elements, or reorder elements using the blue arrows.

### 1.3.3 *Project tree*

In both tabs, BIM model and FDS model, the Project tree is broken into two lists: Libraries and Model. Each of these will be explained in further detail in the appropriate manual sections below.

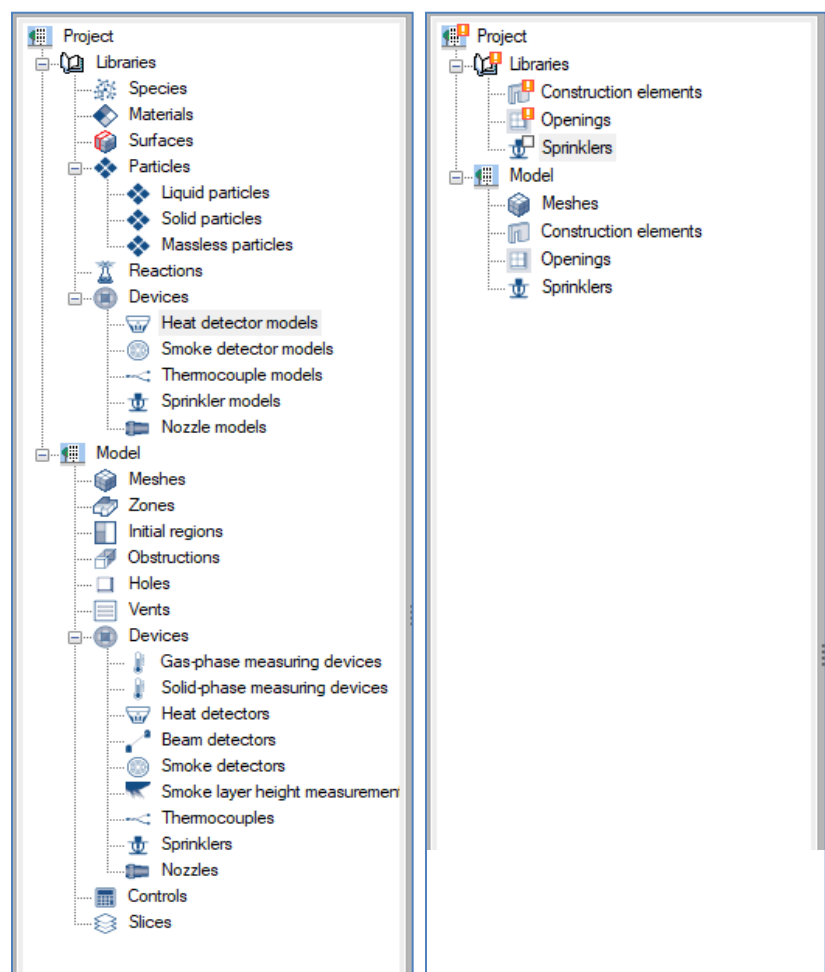


Fig. 5. Project trees ("BIM Model" left, "FDS Model" right)

### 1.3.4 *3D Model window*

The 3D model window allows you to visualize the model in its current state. It is possible to refresh the model to update any changes that have been made, to filter the layers, alter the orientation, and to use slices to display the interior. For navigating the 3D view, the controls are very similar to other CYPE programs. To pan the model, hold down the mouse wheel. To rotate the model, hold the left or right mouse button down. To zoom, simply roll the mouse wheel.

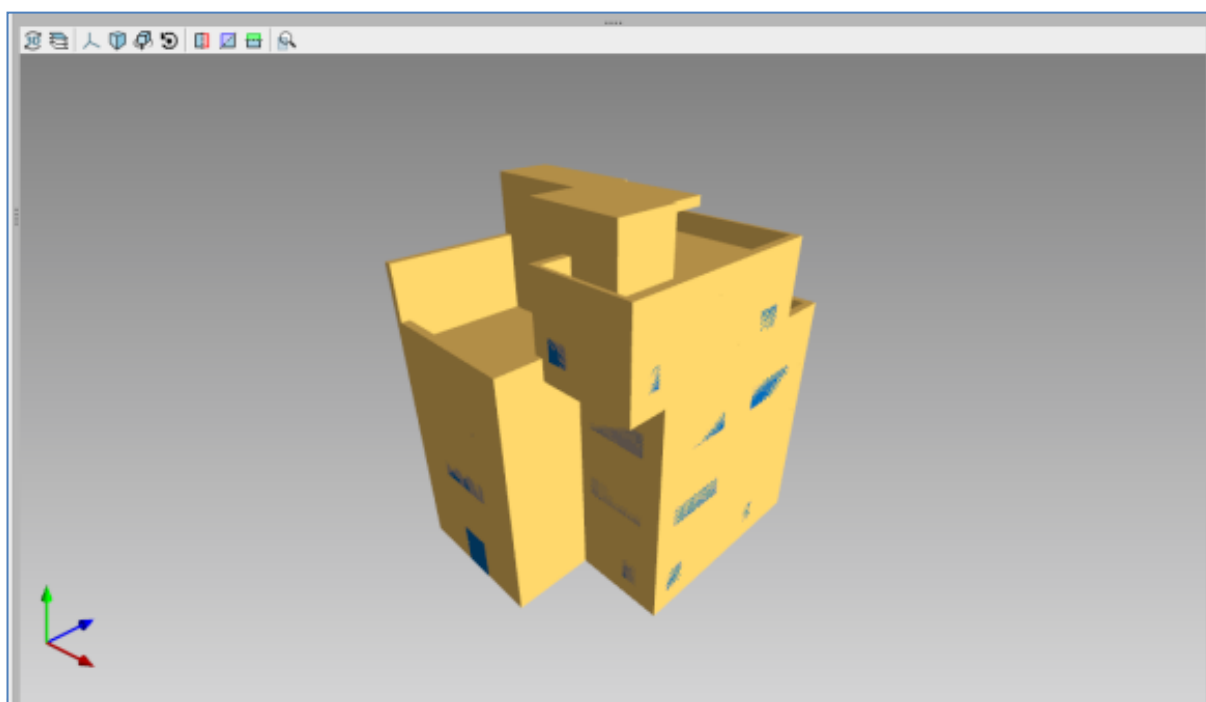


Fig. 6. 3D model

## 2 BIM Model

### 2.1 Libraries

In this section of the tree we find the libraries where the main characteristics of the elements imported from the BIM model are defined: Constructive elements, Gaps and Sprinklers.

#### 2.1.1 *Construction elements*

In the case of linking the project with a BIM model, the construction elements will be shown in this part of the project tree.

When accessing it we can observe in the selection panel all the types of construction elements that exist (facades, interior partitions, forged, defences, etc.). These must be reviewed before exporting to the Model FDS tab. We can know that there are elements to be defined since in the tree we will be indicated with an orange exclamation mark.

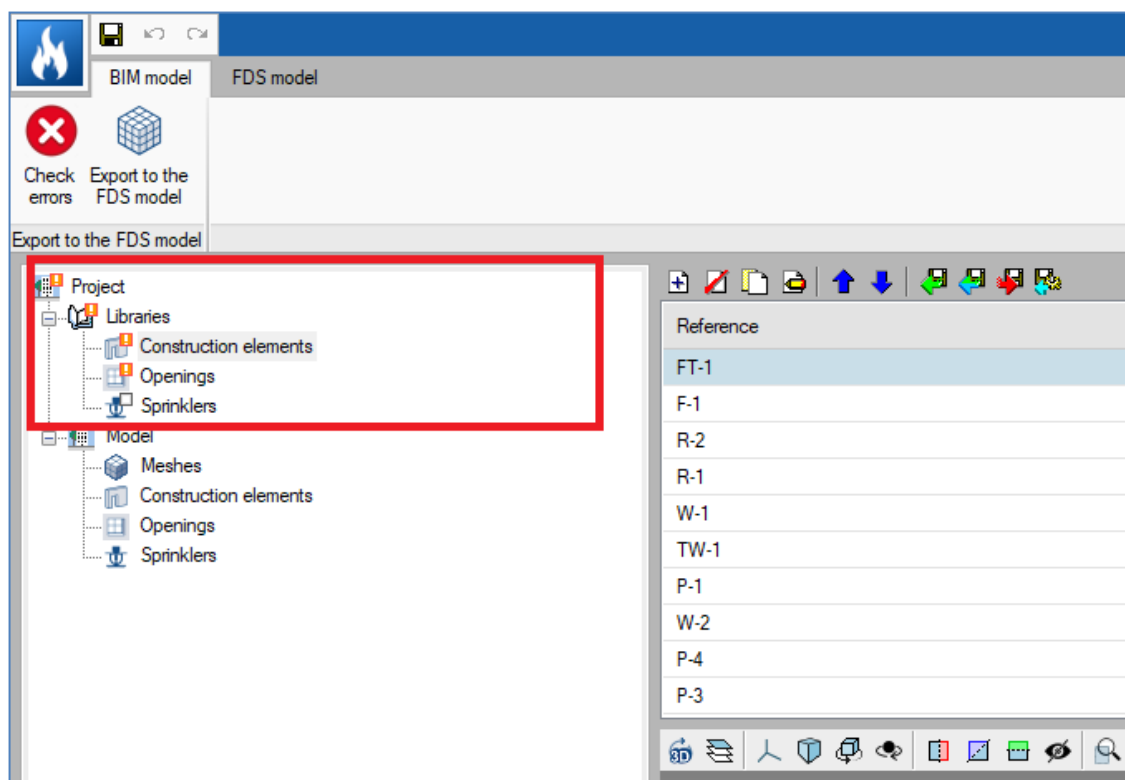


Fig. 7. Construction elements



To define the constructive elements you just have to enter the layers of which the element is formed by defining:

- Layer reference (It will be used as the description of the layer in the FDS Model tab)
- Layer ID (It will be used to describe a particular material of the layer in the FDS Model tab)
- Thickness of the layer

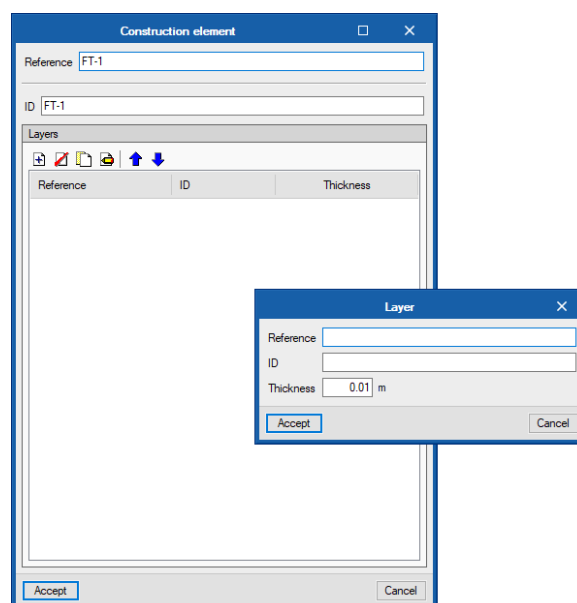


Fig. 8. Construction element prompt

## 2.1.2 *Openings*

The openings are the elements that represent both the doors and glazed openings, and their definition is similar to that of the building elements but the breaking temperature must also be defined for the element. Once this temperature is reached in any of the surfaces of the opening, it disappears from the simulation.

Like the construction elements, an orange exclamation marks the openings whose layers have not been defined.

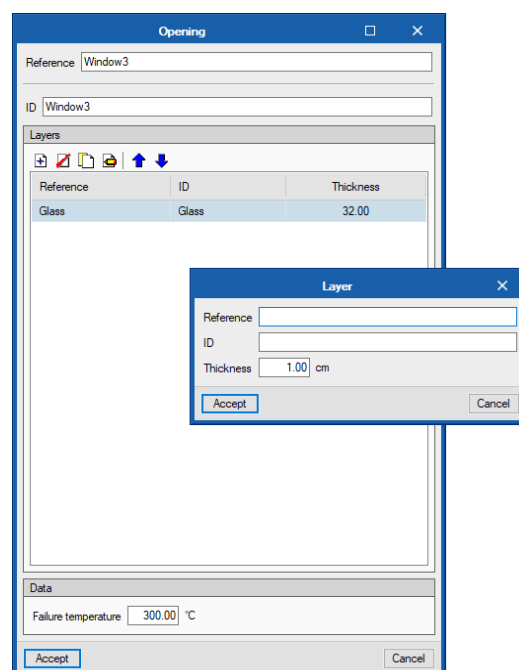


Fig. 9. Opening prompt

### 2.1.3 *Sprinklers*

If there is sprinkler information in the BIM model (information that can be exported from CYPEFIRE Sprinklers), all types of sprinklers present will appear here.

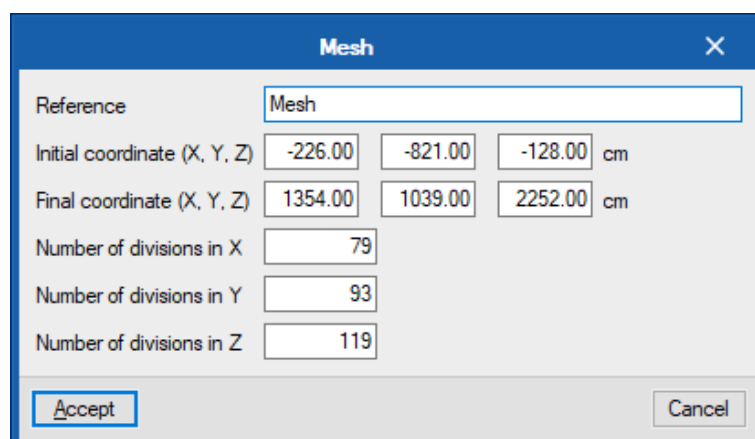
The only information in this section is the reference and the sprinkler ID, with the main properties being defined in the FDS Model tab.

## 2.2 Model

### 2.2.1 *Meshes*

The meshes define the parts of the model in which the simulation is carried out, so all the objects involved in the calculation must be inside a mesh. In the simulation, more than one mesh can be introduced.

Each mesh has its geometric position defined by 3 pairs of coordinates (x, y, z) and divided into uniform cells. The size of the cells is recommended to have the same size in all three directions. By default CYPEFIRE FDS will create a mesh containing the entire model with the number of divisions needed in (x, y, z) to obtain a cell size of 0.20 x 0.20 x 0.20 m.



The image shows a 'Mesh' dialog box with the following fields and values:

Field	Value	Unit
Reference	Mesh	
Initial coordinate (X, Y, Z)	-226.00, -821.00, -128.00	cm
Final coordinate (X, Y, Z)	1354.00, 1039.00, 2252.00	cm
Number of divisions in X	79	
Number of divisions in Y	93	
Number of divisions in Z	119	

Buttons: Accept, Cancel

Fig. 10. Mesh prompt

## 3 FDS Model

### 3.1 General data

#### 3.1.1 *Simulation time*

CYPEFIRE FDS allows the configuration of the following parameters related to the simulation time:

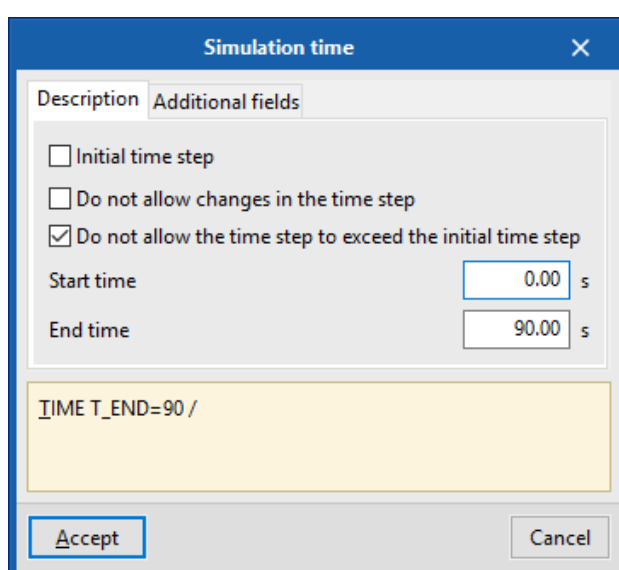


Fig. 11. Simulation time settings

- **Start time.** Allows the indication of the moment in which the results of the simulation will begin to be written into the output files of the FDS calculation engine
- **End time.** Indicates the total duration of the simulation. If the value is 0, it only generates the initial disposition of the model, which allows a quick verification of the geometry in the Smokeview window.
- **Initial time step.** It is possible to indicate the initial time step of the simulation. By default, this value is calculated automatically by dividing the size of a grid cell by the characteristic flow rate. During the calculation, the time step is adjusted so that the condition of CFL (Courant, Friedrichs, Lewy) is fulfilled. The FDS calculation engine uses the following formula to obtain the value of the time step:

$$DT = \frac{5(\delta_x \delta_y \delta_z)^{\frac{1}{3}}}{\sqrt{gH}}$$

Where:

$DT$ : Time steps [s]

$\delta_x$ : Dimension in X of the smallest mesh cell

$\delta_y$ : Dimension in Y of the smallest mesh cell

$\delta_z$ : Dimension in Z of the smallest mesh cell

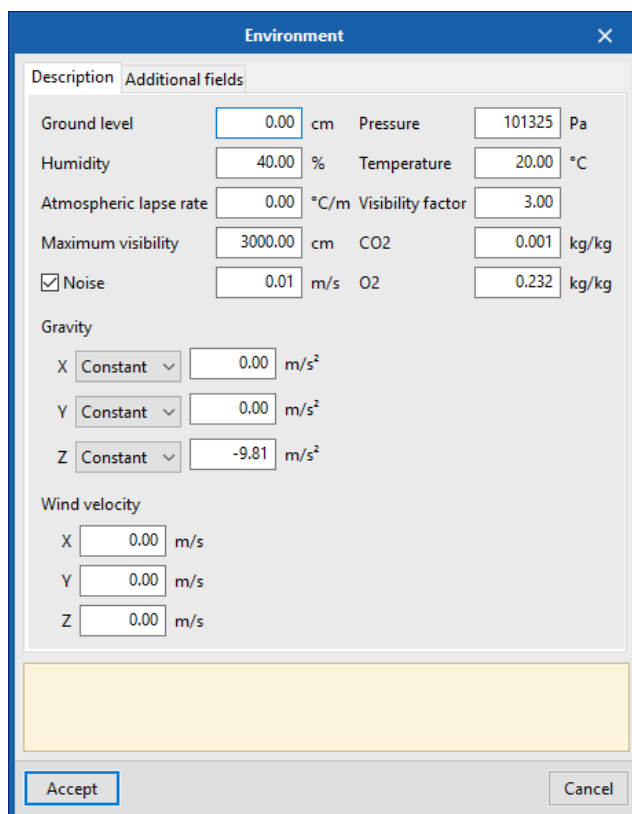
$g$ : Acceleration of gravity

$H$ : Height of the computational domain

- **Do not allow changes in the time step.** Checking this option prevents the FDS calculation engine from automatically adjusting the time step.
- **Do not allow the time step to exceed the initial time step.** By default, the time step can never exceed the initial value. To allow this to happen it is possible to uncheck this option.

### 3.1.2 *Environment*

CYPEFIRE FDS allows the configuration of the following parameters related to the ambient environment in which the simulation is conducted:



The dialog box titled "Environment" contains the following settings:

Description	Additional fields
Ground level	0.00 cm
Humidity	40.00 %
Atmospheric lapse rate	0.00 °C/m
Maximum visibility	3000.00 cm
<input checked="" type="checkbox"/> Noise	0.01 m/s
Pressure	101325 Pa
Temperature	20.00 °C
Visibility factor	3.00
CO2	0.001 kg/kg
O2	0.232 kg/kg

**Gravity**

Direction	Value	Unit
X	Constant 0.00	m/s <sup>2</sup>
Y	Constant 0.00	m/s <sup>2</sup>
Z	Constant -9.81	m/s <sup>2</sup>

**Wind velocity**

Direction	Value	Unit
X	0.00	m/s
Y	0.00	m/s
Z	0.00	m/s

Buttons: Accept, Cancel

Fig. 12. Environment settings

- **Ground level** - Indicates the value of the height above ground level in the simulation.
- **Humidity** - Indicates the value of the relative humidity of the water vapour found in the environment.
- **Atmospheric lapse rate** - Indicates the variation of the ambient temperature according to the height. A negative value indicates that the temperature drops with height.
- **Maximum visibility** - Indicates the maximum visibility value through smoke that can be determined by the FDS calculation engine. The need to establish this value is due to the inability, on the part of FDS, to report an infinite value of visibility.
- **Noise** - When activating this option, the FDS calculation engine initializes the flow field with a very small amount of "noise" to avoid the development of a perfectly symmetrical flow when the limit and the initial conditions are perfectly symmetrical.
- **Pressure** - Indicates the ambient pressure at ground level.
- **Temperature** - Indicates the temperature at the beginning of the simulation.
- **Visibility factor** - Indicates the value of the constant used when calculating the visibility across the smoke.
- **CO2** - Fraction of mass of carbon dioxide found in the environment.
- **O2** - Fraction of oxygen mass found in the environment.
- **Gravity** - Indicates the three components of gravity. The value of the components of gravity can be constant or as a function of time or position.
- **Wind speed** - Indicates the wind speed in each direction at the beginning of the simulation.

### 3.1.3 *Radiation*

It is possible to configure the parameters used to carry out the calculation of the radiation heat transmission equation. You can also disable the calculation of radiation, this saves approximately 20% of CPU time.

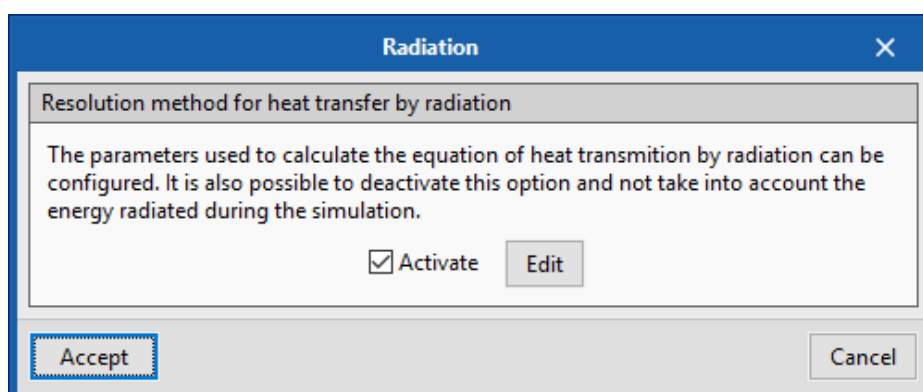
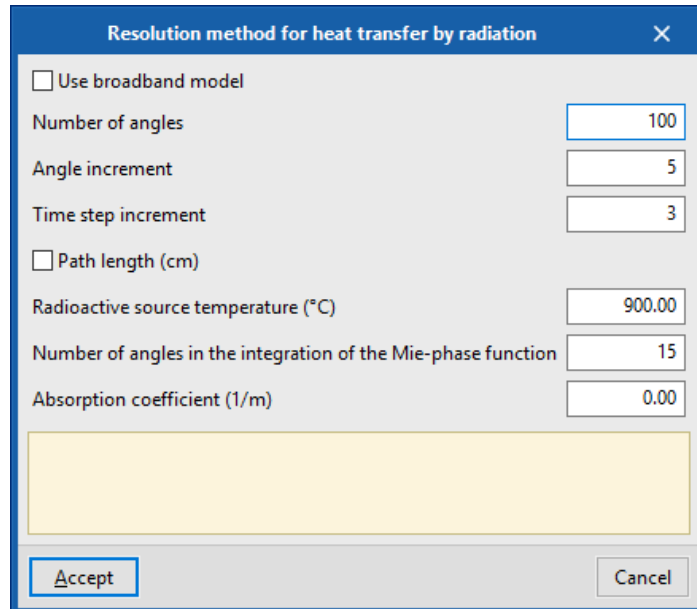


Fig. 13. Radiation prompt

CYPEFIRE FDS permits the configuration of the following parameters related to the calculation of radiation:

- **Use broadband model** - When activated, the radiation calculator will use the broadband model instead of the grey gas model used by default.
- **Number of angles**
- **Angle increment** - Indicates the increment over which the angles must be updated.
- **Time step increment** - It indicates how often FDS must call the calculator of transmission of heat by radiation.
- **Path length** - It is used to determine the range of wavelengths over which the effective absorption coefficients will be calculated. If not defined manually, FDS will default to a step length value equal to five times the size of a mesh cell.
- **Radioactive source temperature** - Indicates the value of the assumed temperature for the radiation source. It is used in the spectral weighting, during the calculation of the average cross sections of dispersion and absorption.
- **Number of angles in the integration of the Mie-phase function** - Indicates the number of angles used in the numerical integration of the Mie function. Increasing this value will obtain a greater precision in the radioactive properties of the water particles.

- **Absorption coefficient** - By activating this option it is possible to establish a constant absorption coefficient that will be used in simulations without combustion or species that radiate.



The dialog box titled "Resolution method for heat transfer by radiation" contains the following settings:

- ☐ Use broadband model
- Number of angles: 100
- Angle increment: 5
- Time step increment: 3
- ☐ Path length (cm)
- Radioactive source temperature (°C): 900.00
- Number of angles in the integration of the Mie-phase function: 15
- Absorption coefficient (1/m): 0.00

At the bottom, there is a large empty yellow rectangular area and two buttons: "Accept" and "Cancel".

Fig. 14. Radiation settings

### 3.1.4 *Output parameters*

CYPEFIRE FDS allows you to configure the following parameters related to the output files of the simulation:

- Number of data downloads in the output file, for each calculation.
- Maximum number of Lagrangian particles. Indicates the maximum number of Lagrangian particles that can be included simultaneously in a mesh.
- Delete diagnosis. Activating this option will reduce the level of detail of information about the status of the simulation shown in the output file with extension ".out".
- Generate mass file of gaseous species. When activating this option, an output list will be created with the total mass of all the species as a function of time. It must be borne in mind that performing this calculation will considerably increase the time necessary to perform the simulation.
- Generate smoke and fire animation. Activating this option will produce an animation of the smoke and fire that will be displayed during the simulation display in Smokeview.

- **Speed error file.** When activating this option, a file will be created with the maximum error associated to the normal component of the speed in the solid or interpolated limits.
- **Error file.** Activating this option will generate an output file with extension ".notready" that will be deleted if the simulation is completed successfully.
- **Periodically delete the temporary output files.** When activating this option, FDS will periodically clean the temporary output files and write the data in the respective result files. In this way you can easily visualize the model in Smokeview while the simulation is being performed.
- **Precision of the output file.** Indicates the number of significant digits of the mantissa and the exponent in the numerical data entered in FDS.
- **Intervals of data writing in the output file.**
- **Limit the maximum number of columns.** When activating this option, the maximum number of columns of the files with extension .csv will be limited where the data of the devices (DEVC) and controllers (CTRL) are displayed.

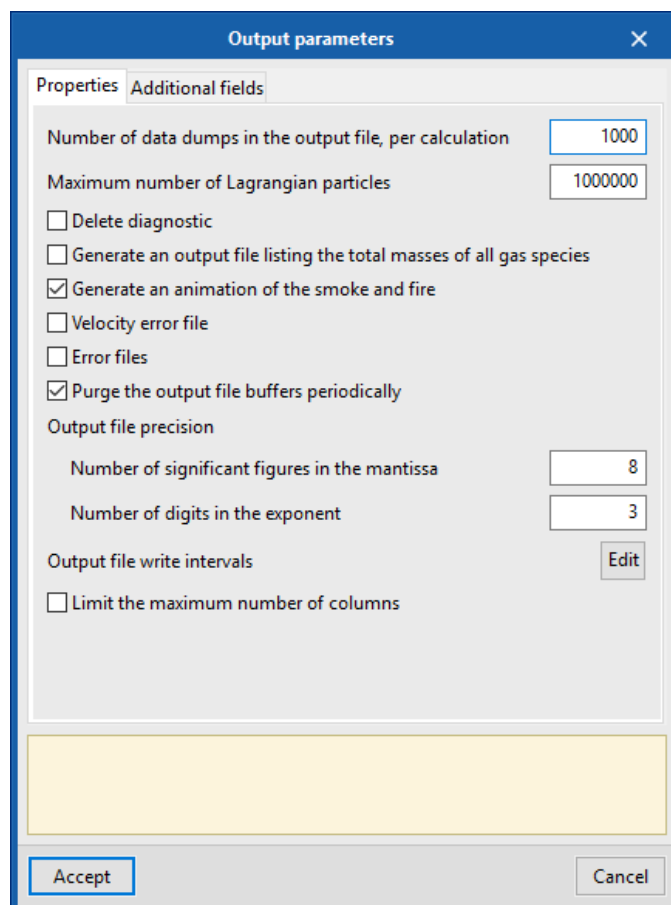


Fig. 15. Output parameters



## 3.2 Species

Gaseous species can be used as reactive species of the combustion model or as non-reactive species in the simulation of air flow.

It is possible to indicate the initial concentration of a predefined species in the simulation through the initial mass fraction value of the gas, in the species section of the FDS model. The FDS calculation engine creates by default an Air species that is defined as a compound of  $N_2$ ,  $O_2$ ,  $CO_2$  and  $H_2O$ . This species fills the space of the initial simulation environment that has not been occupied by the rest of the species.

Species can be introduced into the simulation through the emission parameters of the surfaces.

Reference	Initial mass fraction (kg/kg)	FDS command
ACETONE	0.000	&SPEC ID='ACETONE' /
ACETYLENE	0.000	&SPEC ID='ACETYLENE' /
ACROLEIN	0.000	&SPEC ID='ACROLEIN' /
AMMONIA	0.000	&SPEC ID='AMMONIA' /
ARGON	0.000	&SPEC ID='ARGON' /
BENZENE	0.000	&SPEC ID='BENZENE' /
BUTANE	0.000	&SPEC ID='BUTANE' /
CARBON	0.000	&SPEC ID='CARBON' /
CARBON DIOXIDE	0.000	&SPEC ID='CARBON DIOXIDE' /
CARBON MONOXIDE	0.000	&SPEC ID='CARBON MONOXIDE' /
CHLORINE	0.000	&SPEC ID='CHLORINE' /
DODECANE	0.000	&SPEC ID='DODECANE' /

Fig. 16. Species selection window

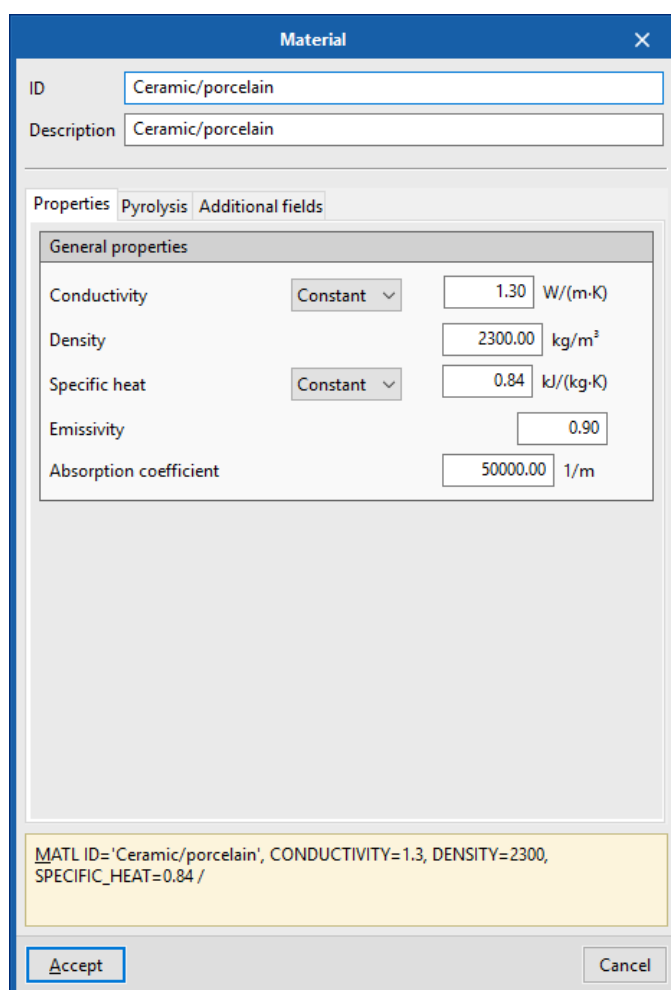
## 3.3 Materials

The solid objects that make up the geometric model are often made up of several layers of different materials. Therefore, to carry out the simulation, the thermal properties of each of these materials must be specified. In the same way, it is also possible to specify the characteristics of the liquid materials found in the scenario, which are used to represent types of fuel.

### 3.3.1 *Properties*

Within the Properties tab of the Materials panel, it is possible to specify the following parameters related to the thermal characteristics of the material:

- **Conductivity.** It can be defined as a function of temperature by means of a ramp function.
- Density.
- **Specific heat.** It can be defined as a function of temperature by means of a ramp function.
- **Emissivity.** Indicates the fraction of thermal radiation emitted by the material. Its value must be between 0 and 1.
- **Absorption coefficient.** Indicates the depth over which the material can absorb thermal radiation.



The image shows a 'Material' dialog box with the following fields and values:

Field	Value	Unit
ID	Ceramic/porcelain	
Description	Ceramic/porcelain	
<b>General properties</b>		
Conductivity	Constant	1.30 W/(m·K)
Density		2300.00 kg/m³
Specific heat	Constant	0.84 kJ/(kg·K)
Emissivity		0.90
Absorption coefficient		50000.00 1/m

Summary text at the bottom: MATL ID='Ceramic/porcelain', CONDUCTIVITY=1.3, DENSITY=2300, SPECIFIC\_HEAT=0.84 /

Fig. 17. Material, properties

### 3.3.2 *Pyrolysis*

The pyrolysis model of the material describes the reactions that occur during its combustion process and the products that are generated.

The pyrolysis properties concerning solid and liquid materials are different. Consequently, the type of material must be defined before specifying its characteristics.

#### 3.3.2.1 *Solid materials*

Solid materials can have several associated chemical reactions without the need for them to occur at the same temperature.

The Pyrolysis tab of CYPEFIRE FDS allows you to configure the following general material options:

- **Allow contraction of the material.** When activating this option, to conserve the mass, the thickness of the material will be reduced after a reaction process in which the products have a higher density than the original material.
- **Allow the expansion of the material.** When activating this option, to conserve the mass, the thickness of the material will increase after a reaction process in which the products have a lower density than the original material.

By adding a reaction to the material you must specify its kinetic properties and its products.

The equation that FDS uses to determine the speed of the reaction is the following:

$$r_{ij} = A_{ij} Y_{s,i}^{n_{s,ij}} \exp\left(-\frac{E_{ij}}{RT_s}\right) X_{O_2}^{n_{O_2,ij}}; \quad Y_{s,i} = \left(\frac{\rho_{s,i}}{\rho_s(0)}\right)$$

Where:

- $r_{ij}$ : Speed of reaction to temperature  $T_s$
- $\rho_{s,i}$ : Material density [kg/m<sup>3</sup>]
- $\rho_s(0)$ : Initial density of the layer [kg/m<sup>3</sup>]
- $n_{s,ij}$ : Order of reaction
- $A_{ij}$ : Pre-exponential factor [s<sup>-1</sup>]
- $E_{ij}$ : Activation energy [kJ/kmol]
- $X_{O_2}$ : Local fraction of oxygen volume
- $R$ : Universal constant of ideal gases (8,3143 J\*K<sup>-1</sup>\*mol<sup>-1</sup>)

The user can choose between manually entering the values of the pre-exponential factor and the activation energy or calculating them from the following equation:

$$E_{i,j} = \frac{er_{p,i}}{Y_{s,i}(0)} \frac{RT_{p,i}^2}{\dot{T}} \quad ; \quad A_{i,j} = \frac{er_{p,i}}{Y_{s,i}(0)} e^{E/RT_{p,i}}$$

Where:

$T_{p,i}$ : Reference temperature [°C]

$r_{p,i}/Y_{s,i}(0)$ : Reference velocity [ $s^{-1}$ ]

$\dot{T}$ : Heat rate [°C/min]

In both cases the value of the reaction order must be specified,  $n_s$ .

In cases where the value of the reference temperature is not available, it can be approximated from the pyrolysis range:

$$\frac{r_{p,i}}{Y_{s,i}(0)} = \frac{2\dot{T}}{\Delta T} (1 - v_{s,i})$$

Where:

$\Delta T$ : Pyrolysis range [°C]

$v_{s,i}$ : Production of solid waste

To end up with the kinetic parameters, the program modifies the equation to define pyrolysis models that do not follow the Arrhenius function.

$$r_{ij} = A_{ij} Y_{s,i}^{n_{s,ij}} \exp\left(-\frac{E_{ij}}{RT_s}\right) \max[0, S_{thr,ij}(T_s - T_{thr,ij})]^{n_{t,ij}}$$

Where:

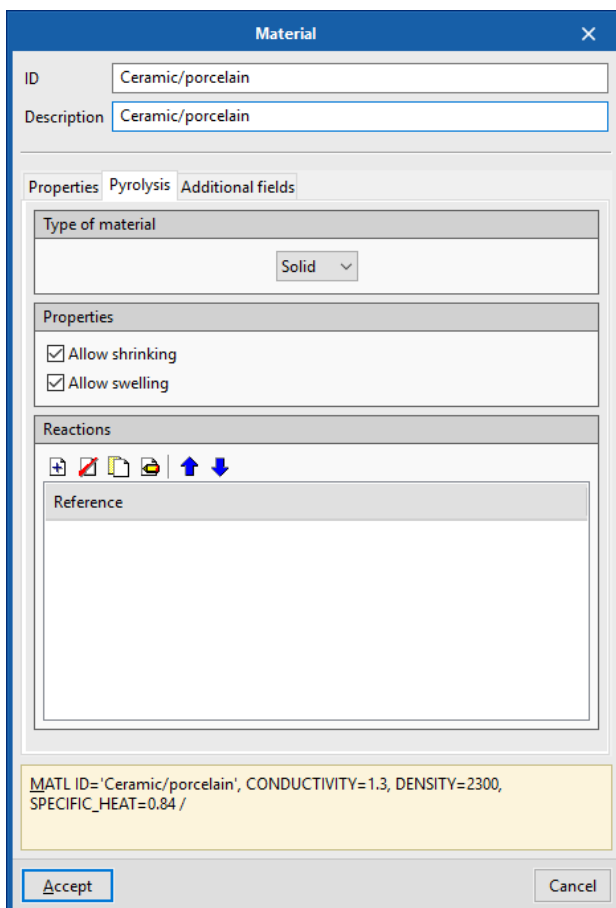
$T_{thr,ij}$ : Threshold temperature [°C]

In this case it will be necessary to enter the values of the threshold temperature,  $T_{thr}$ , and the exponent of the reaction,  $n_t$ .

Once the kinetic parameters that govern the reaction have been defined, the products generated after the combustion and their quantity can be specified in the Products tab. These can be both chemical species and other materials previously defined in the project.

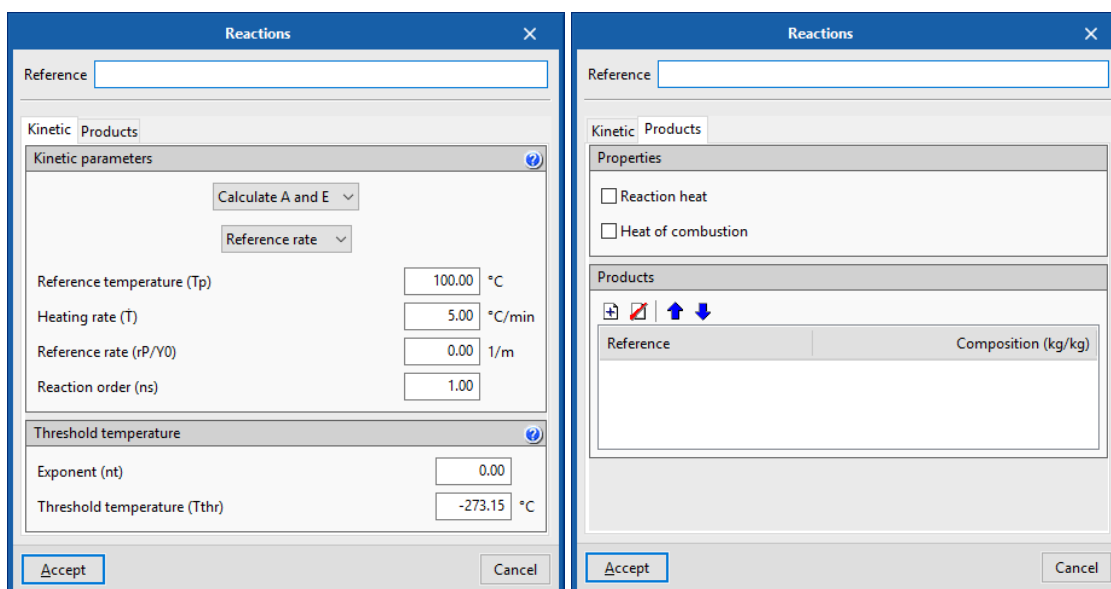
Finally, it is possible to specify the heat values of the reaction, which indicates the amount of energy per unit mass of the reactant that is consumed in the reaction, and the heat of combustion that represents the amount of energy emitted per unit mass of the material when subjected to a complete combustion in the presence of oxygen.

It must be taken into account that the FDS calculation engine allows each solid material to have a maximum of 10 associated reactions.



The 'Material' dialog box is shown with the 'Pyrolysis' tab selected. The 'ID' and 'Description' fields both contain 'Ceramic/porcelain'. Under the 'Properties' section, 'Type of material' is set to 'Solid'. The 'Allow shrinking' and 'Allow swelling' checkboxes are both checked. The 'Reactions' section is empty, showing a list of references. At the bottom, a status bar displays: 'MATL ID= 'Ceramic/porcelain', CONDUCTIVITY=1.3, DENSITY=2300, SPECIFIC\_HEAT=0.84 /'. 'Accept' and 'Cancel' buttons are at the bottom.

Fig. 18. Material, pyrolysis



Two 'Reactions' dialog boxes are shown side-by-side. The left dialog box has the 'Kinetic' tab selected, showing 'Kinetic parameters' with fields for 'Reference temperature (Tp)' (100.00 °C), 'Heating rate (T)' (5.00 °C/min), 'Reference rate (rP/Y0)' (0.00 1/m), 'Reaction order (ns)' (1.00), 'Exponent (nt)' (0.00), and 'Threshold temperature (Tthr)' (-273.15 °C). The right dialog box has the 'Products' tab selected, showing 'Properties' with 'Reaction heat' and 'Heat of combustion' checkboxes, and a 'Products' section with a table for 'Reference' and 'Composition (kg/kg)'. Both dialogs have 'Accept' and 'Cancel' buttons at the bottom.

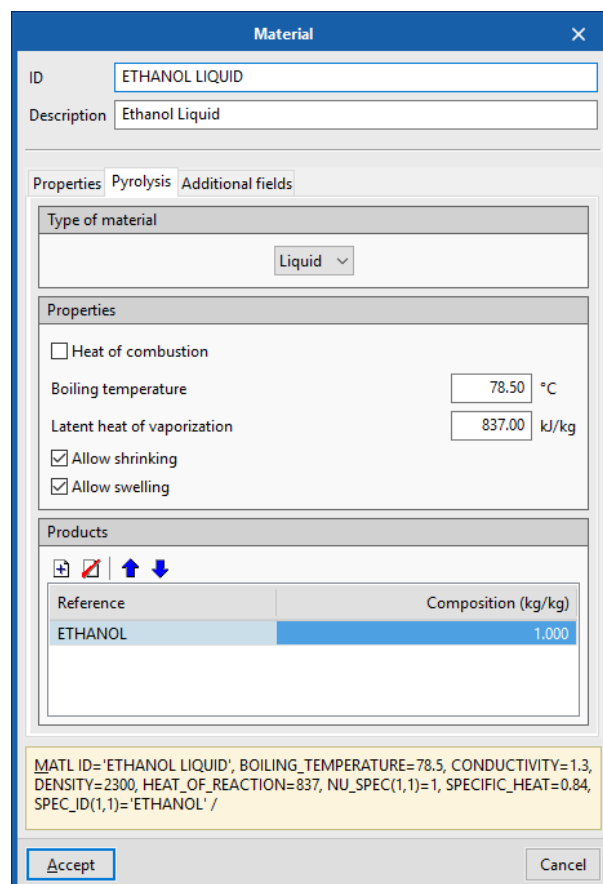
Fig. 19. Reaction, kinetic parameters and products

### 3.3.2.2 *Liquid materials*

Unlike solid materials, liquid fuels can only have an associated reaction.

It is possible to specify the following general properties:

- **Heat of combustion.** It is the amount of energy emitted per unit mass of the material when subjected to a complete combustion in the presence of oxygen.
- **Boiling temperature.** Temperature at which the vapour pressure of the liquid equals the vapour pressure of the medium in which it is located.
- **Latent heat of vaporization.** Energy required by the material to change from liquid to gaseous phase.
- **Allow contraction of the material.** When activating this option, to conserve the mass, the thickness of the material will be reduced after a reaction process in which the products have a higher density than the original material.
- **Allow the expansion of the material.** When activating this option, to conserve the mass, the thickness of the material will increase after a reaction process in which the products have a lower density than the original material.



The screenshot shows the 'Material' dialog box with the following details:

- ID:** ETHANOL LIQUID
- Description:** Ethanol Liquid
- Properties tab:**
  - Type of material:** Liquid (dropdown menu)
  - Properties:**
    - ☐ Heat of combustion
    - Boiling temperature:** 78.50 °C
    - Latent heat of vaporization:** 837.00 kJ/kg
    - ☒ Allow shrinking
    - ☒ Allow swelling
  - Products:**

Reference	Composition (kg/kg)
ETHANOL	1.000
- Footer:**

MATL ID='ETHANOL LIQUID', BOILING\_TEMPERATURE=78.5, CONDUCTIVITY=1.3, DENSITY=2300, HEAT\_OF\_REACTION=837, NU\_SPEC(1,1)=1, SPECIFIC\_HEAT=0.84, SPEC\_ID(1,1)='ETHANOL' /

Buttons: Accept, Cancel

Fig. 20. Liquid material, pyrolysis

In terms of the products generated by the reaction, they can be both species and other materials, in the same way as in the reactions of solid materials.

## 3.4 Surfaces

The surfaces are used with the purpose of being able to specify the boundary conditions of the different solid elements or openings located inside the computational domain.

The surfaces in CYPEFIRE FDS must have a unique reference and a colour. It is also possible to indicate if it is adiabatic or, if not, to specify all the properties that define its behaviour. To speed up this task, the Surface panel is structured based on the following tabs:

- General properties
- Thermal properties
- Pyrolysis
- Issue
- Additional fields

### 3.4.1 *Default surfaces*

There are certain predefined surfaces within the FDS calculation engine, which have particular characteristics. Consequently, when starting a new project in CYPEFIRE FDS these surfaces will already be specified and it will not be possible to eliminate them.

- **INERT.** Represents an inert wall at room temperature and is the default boundary condition for all solid surfaces.
- **OPEN.** It is only used to indicate that there is an opening to the outside in the limits of a mesh.
- **MIRROR.** Represents a plane of symmetry. It is a surface without its own flow, which allows a free movement and inverts the flow.
- **PERIODIC.** This type of surface is used to establish periodic limits.

The surfaces of type OPEN, MIRROR and PERIODIC can only be assigned to entities of type VENT (Vents) and these can only be located in the outer limits of a mesh.

### 3.4.2 *General properties*

Within this group of parameters are the characteristics that define the general behaviour of the surface.

First, it is possible to indicate the geometry of the surface. The solid objects introduced in FDS are adjusted to the rectilinear geometry of the mesh; consequently, all physical entities are represented as "boxes". The geometry of the surface can be used to describe the thermal characteristics of spherical or cylindrical objects in the obstructions where it is applied. In addition, in case of using the surface as a particle, the different dimensions of the geometry should be defined according to its type. The geometry classes available in CYPEFIRE FDS are:

- Cartesian.
- Spherical.
- Cylindrical.

CYPEFIRE FDS also allows the calculation engine to indicate that a surface is used as an escape path between two pressure zones. When activating this option, you must indicate the two pressure zones, previously defined, to which reference is made.

The surfaces that make up the solid objects can be formed by several layers and these, in turn, can be composed of several materials. When adding a new layer to the surface, in addition to listing the materials that make it up, you must establish a value for its thickness and temperature.

Once the layers have been defined, CYPEFIRE FDS includes the option 'Separation of layers' which allows specifying the number of layers involved in the ejection of fuel vapour to the outside of the surface. By default, the layer separation is 0.5 times the number of layers for surfaces with exposed back, and equal to the number of layers for the rest.



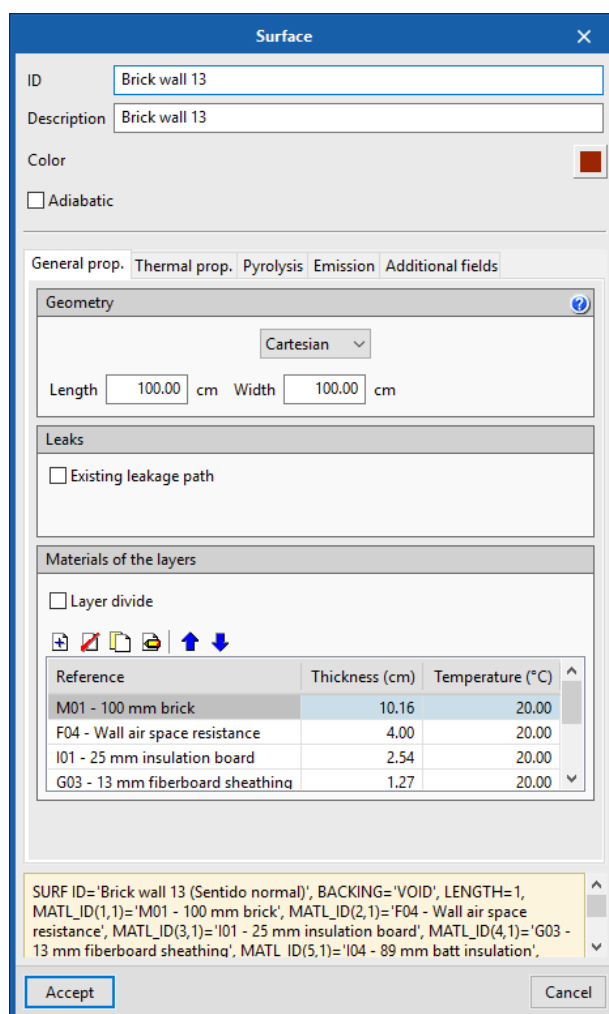


Fig. 21. Surface, general properties

### 3.4.3 *Thermal properties*

The model of convection heat transmission that FDS uses in the simulation can be modified in several ways. Below are the options that CYPEFIRE FDS allows us to use:

- **Default convection heat transfer model**

- In the LES calculation, the heat transfer coefficient  $h$ , measured in  $W/(m^2 \cdot K)$ , is determined from the following equation:

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

Where  $C$  is an empirical coefficient for natural convection,  $L$  is a characteristic length related to the size of the physical obstruction,  $k$  is the thermal conductivity of the gas and  $Nu$  is the Nusselt number.

- **Logarithmic law of the wall**

- The local coefficient of heat transfer will be obtained from the following equation:

$$h = \frac{\dot{q}_w^n}{T_g - T_w} = \frac{\rho_w * c_p * u_\tau}{T^+}$$

- **Specify the heat transfer coefficient**

- A constant value of the convection heat transfer coefficient can be directly indicated.

- **Specify the heat flow on the solid surface**

- Instead of altering the heat transfer coefficient by convection, a fixed heat flow can be specified directly. There are two ways to do it:
  - Indicate the value of the net heat flow. FDS will calculate the surface temperature necessary to ensure that the combined heat flux by radiation and convection is equal to the net heat flux.
  - Indicate the flow of heat by convection and the heat flow by radiation separately. The heat flux by radiation will be determined from the emissivity and the temperature of the surface.

The reverse side of a surface represents the boundary condition "behind" it. CYPEFIRE FDS allows the selection of the following types of reverse:

- **Empty.** It is assumed that the back of the wall is in an air space. The temperature of this zone can be defined by the temperature option on the back.
- **Exposed.** It will allow the surface to transmit heat to the space behind the wall. This option will only work if the wall is less than or equal to the thickness of a grid cell and if there is volume of the computational domain on the other side of the wall.
- **Isolated.** It is assumed that the wall is behind an insulating material, in which case there will be no loss of heat to the material on the back.

In addition to the type of reverse, it is also possible to indicate a value for its emissivity.

The last group of options on this tab is related to the temperature. It is possible to indicate the temperature of the outer limit of the surface which is used, together with the emissivity, to determine the heat flux by radiation, when the total heat flow is specified in

the radiation heat transmission options. The temperature of the surface may not be constant and vary its value in time, to indicate that this occurs the ramp function option is included.

In the case that a reverse type of empty CYPEFIRE FDS has been selected, it is also possible to set the temperature of the air space where the reverse side of the surface is located.

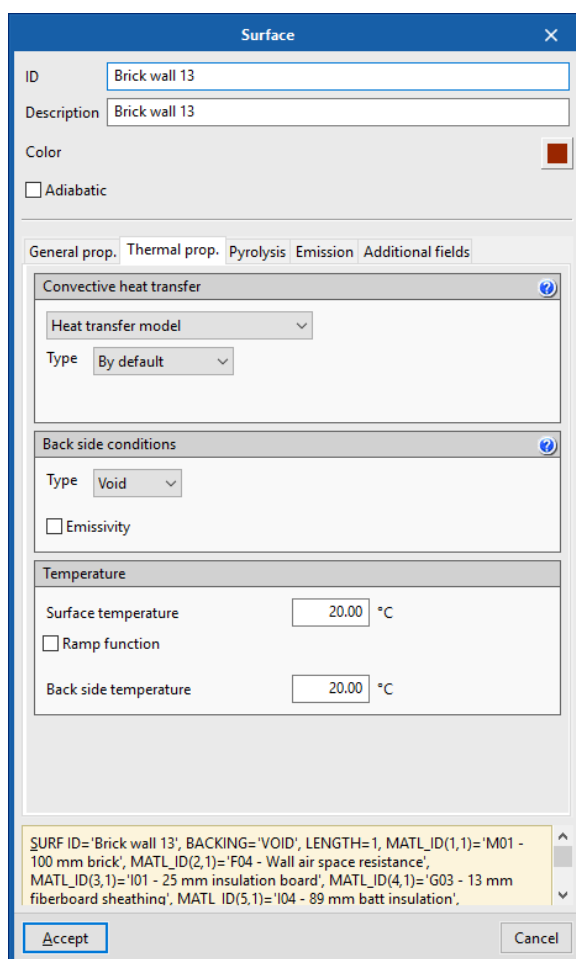


Fig. 22. Surface, thermal properties

### 3.4.4 *Pyrolysis*

Within this tab it is possible to detail the pyrolysis model that is carried out on the surface. This can be defined by each of the parameters specified in the definition of the materials that make up the surface or, on the contrary, you can establish specific properties. In both cases, FDS can indicate if it is desired that the solid objects where the surface is used disappear once they have been consumed. The objects will be eliminated from the simulation cell by cell, as a consequence of the mass contained within each solid cell being consumed by the pyrolysis reactions or by the defined heat emission rate.

In the case of choosing the manual definition of the pyrolysis model, the following parameters related to the emission of heat and the ignition of the surface can be defined:

- **Emission of heat per unit area.**
- **Speed of mass loss.**
- **Ramp function.** It allows describing non-constant behaviour, as a function of time, of the heat emission from the surface.
- **Coefficient E.** Empirical constant that depends on the properties of solid fuel material and its geometrical configuration. It is used to calculate the reduction of the combustion speed as a result of the presence of water.
- **Ignition temperature.** Indicates the temperature from which the object will begin to burn.
- **Vaporization heat.** Indicates the amount of energy needed to burn the fuel.

When specifying the heat emission from the surface, you must choose between entering the value of the heat emission per unit area or the speed of mass loss.

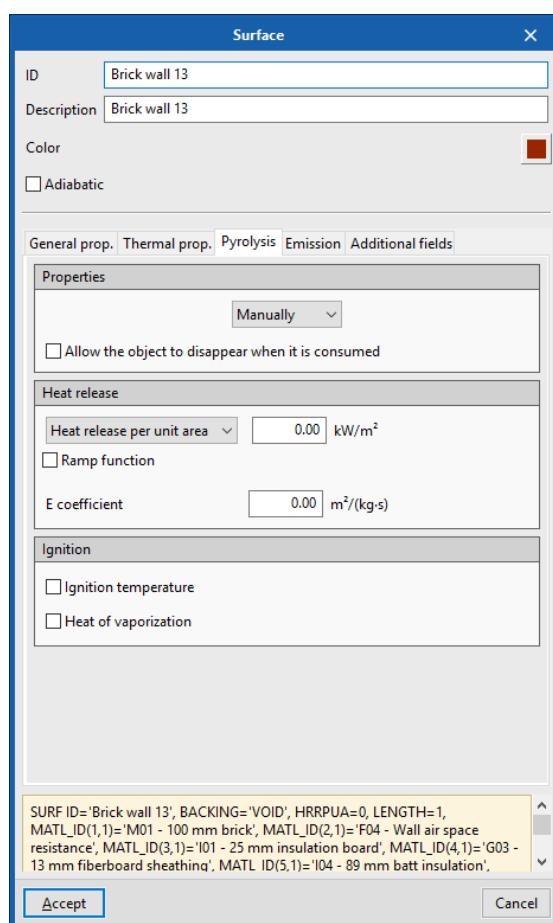


Fig. 23. Surface, pyrolysis

### 3.4.5 *Emission*

By default, gaseous species do not penetrate solid surfaces. However, if it is desired to indicate the existence of a species emission flow, FDS provides two ways of doing so:

- If the *mass fraction* of the species must have a known value within the flow, this value can be indicated together with the *flow velocity*, the *volumetric flow* or the *total mass flow*.
- In case you want to represent the emission of species from your mass flow, you can indicate this value and it will not be necessary to specify the *flow velocity*, the *volumetric flow* or the *total mass flow*.

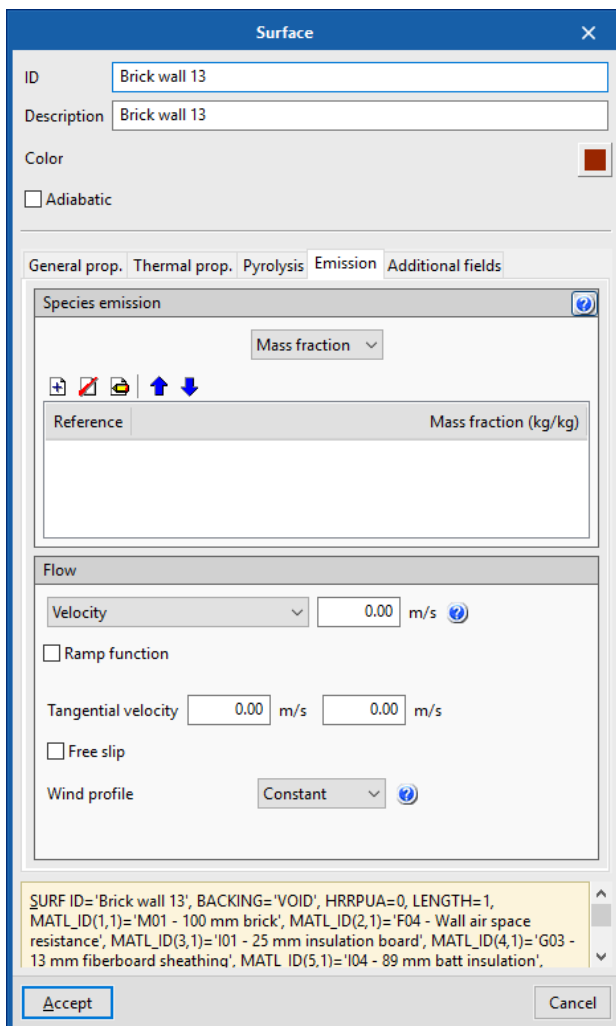
The options that CYPEFIRE FDS includes to define the characteristics of the flow are the following:

- **Speed.** The normal component of the speed is indicated by this parameter.
- **Volumetric flow.** Instead of specifying the speed it is possible to indicate the volumetric flow.
- **Total mass flow.** Instead of specifying the speed it is possible to indicate the total mass flow.
- **Ramp function.** It allows describing non-constant behaviour, as a function of time, flow velocity, volumetric flow or total mass flow.
- **Free sliding.** This option allows you to adjust the surface tension to 0 (eliminates viscous friction).
- **Wind profile.** The speed of the wind profile at any outer limit will be constant by default; however, FDS allows generating other profiles:
  - **Parabolic.** It produces a parabolic profile where the speed specified for the flow is the maximum speed.
  - **Atmospheric.** Produces an atmospheric wind profile that follows the following power law:

$$u = u_0(z/z_0)^P$$

For this case, the values of  $z_0$  and  $p$  must be specified. The parameter  $u_0$  corresponds to the speed indicated for the flow while  $z$  is the height of the ground level specified in the *Environment* options.


A negative value of *speed*, *volumetric flow* or *total mass flow* will indicate that the flow is directed into the computational domain while, if it is positive, the flow direction will point to the outside of the domain.



Surface

ID: Brick wall 13

Description: Brick wall 13

Color: 

☐ Adiabatic

General prop. Thermal prop. Pyrolysis Emission Additional fields

Species emission

Mass fraction

Reference Mass fraction (kg/kg)

Flow

Velocity: 0.00 m/s

☐ Ramp function

Tangential velocity: 0.00 m/s 0.00 m/s

☐ Free slip

Wind profile: Constant

SURF ID='Brick wall 13', BACKING='VOID', HRRPUA=0, LENGTH=1, MATL\_ID(1,1)='M01 - 100 mm brick', MATL\_ID(2,1)='F04 - Wall air space resistance', MATL\_ID(3,1)='I01 - 25 mm insulation board', MATL\_ID(4,1)='G03 - 13 mm fiberboard sheathing', MATL\_ID(5,1)='I04 - 89 mm batt insulation'

Accept Cancel

Fig. 24. Surface, emission

## 3.5 Particles

Lagrangian particles can be used to represent a wide variety of objects that are too small to be solved in the numerical grid. The FDS calculation engine supports three classes of Lagrangian particles: "Particles without mass", "Liquid particles" and everything else.

### 3.5.1 *Liquid particles*

#### 3.5.2 *Description*

- **Controller.** FDS can control the presence of liquid particles in the simulation depending on the response of a control or device.
- **Species.** To define a liquid particle, you must specify the species that forms it. If a liquid particle is assigned the species "WATER VAPOR" it will take the thermophysical properties of the water, the radiative properties of the water and it will be coloured blue in the Smokeview viewer.
- **Thermal properties.** The "Heat of combustion" of the particle can be indicated. The drops will evaporate in an amount equivalent to the fuel vapour in such a way that the resulting heat release rate (assuming complete combustion) is equal to the evaporation rate multiplied by the "Heat of combustion".
- **Size distribution.** The size distribution of the liquid particles is specified by a cumulative volume fraction (FVC) indicated in the field "Type of distribution". The default distribution is "Rossin-Rammler-Lognormal":

$$1. \quad F(D) = \begin{cases} \frac{1}{\sqrt{2\pi}} \int_0^D \frac{1}{\sigma D'} \exp\left(-\frac{\left[\ln\left(\frac{D'}{D_{V,0.5}}\right)\right]^2}{2\sigma^2}\right) dD' & (D \leq D_{V,0.5}) \\ 1 - \exp\left(-0.693\left(\frac{D}{D_{V,0.5}}\right)^\gamma\right) & (D > D_{V,0.5}) \end{cases}$$

Where:

$\sigma$ : Amplitude of the Lognormal distribution

$\gamma$ : Amplitude of the Rosin-Rammler distribution

$D_{V,0.5}$ : Mean diameter [ $\mu\text{m}$ ]

Alternatively, it is possible to specify the distributions "Lognormal" or "Rosin-Rammler" alone, instead of the combination of both. In order to avoid the FDS calculation engine generating a size distribution, the type of distribution "Constant" can be indicated, in which case all the particles will have a diameter equal to the "Mean diameter" indicated.

It is also possible to avoid generating infinitely large particles by specifying a "Maximum diameter". Droplets smaller than the "Minimum Diameter" evaporates in a single step of time.

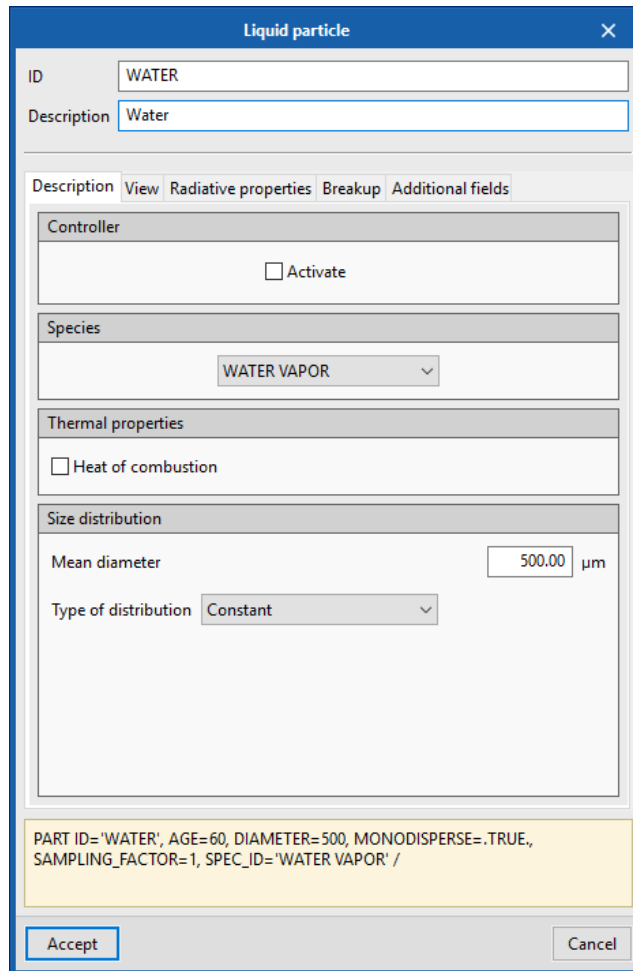


Fig. 25. Liquid particle, description

### 3.5.2.1 *Visualisation*

- **Colour.** It is possible to indicate on the basis of which of the following properties we want the particle to obtain its colour:
  - Temperature
  - Diameter
  - Mass
  - Duration
  - Speed

If no property is specified, Smokeview will draw the particles in a single colour that can also be indicated in this panel. By default, the water will be coloured blue and the rest of the particles black.



- **Number of particles.** It is possible to define the "Sampling Factor" and the "Duration" of the particles. The first parameter is used to reduce the size of the particle output file that is used for the simulation in Smokeview. The default value for particles without mass is 1 (all particles are seen in Smokeview) and for the rest 10. On the other hand, the "Duration" indicates the amount of time in which a particle exists in the simulation.

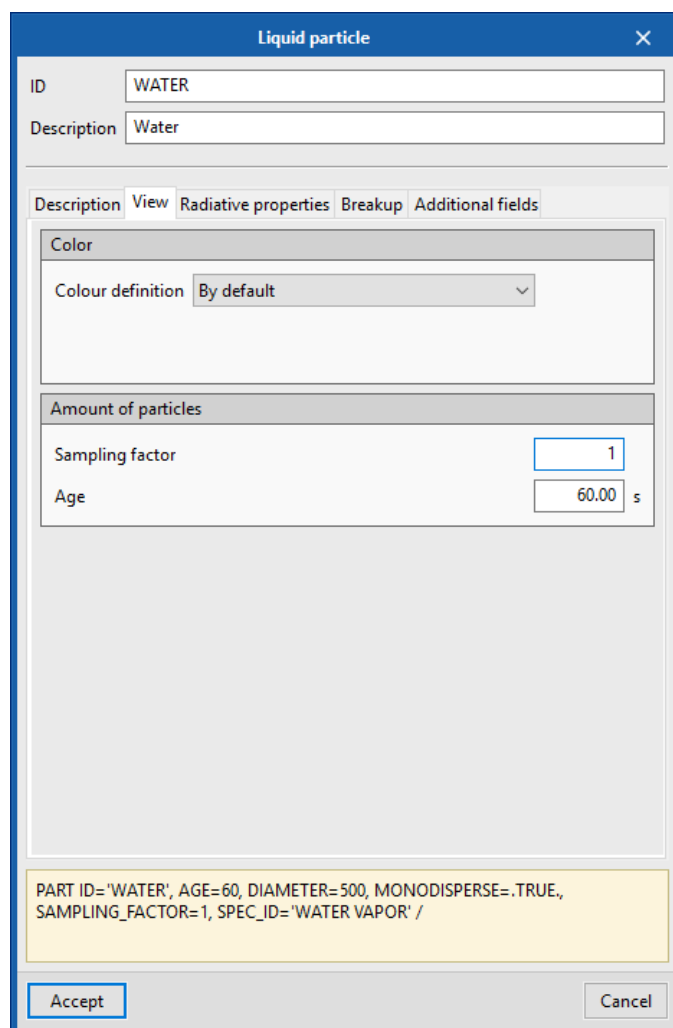


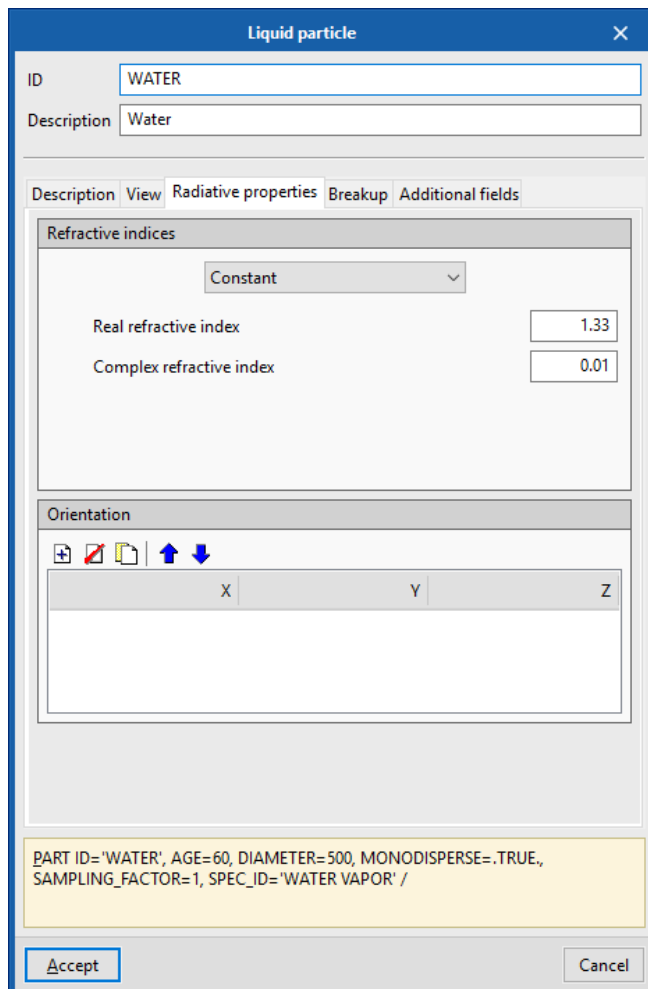
Fig. 26. Liquid particle, view

### 3.5.2.2 *Radiative properties*

The radiative properties of water and fuel are obtained automatically. In the case of fuel, the properties of heptane are assumed.

- **Refractive indexes.** It is possible to directly indicate the value of the "Real Refractive Index" and the "Complex Refractive Index" in case these are considered constant. Alternatively, values of these two parameters can be indicated as a function of the wavelength.

- **Orientation.** In case the radiation heat flux is not evenly distributed around the particle, it is possible to divide the particle into pieces, each with its own orientation.



ID: WATER  
 Description: Water

Description View Radiative properties Breakup Additional fields

Refractive indices  
 Constant  
 Real refractive index: 1.33  
 Complex refractive index: 0.01

Orientation  
 X Y Z

PART ID='WATER', AGE=60, DIAMETER=500, MONODISPERSE=.TRUE., SAMPLING\_FACTOR=1, SPEC\_ID='WATER VAPOR' /

Accept Cancel

Fig. 27. Liquid particle, radiative properties

### 3.5.2.3 *Disintegration*

It is possible to indicate that the particles may undergo secondary disintegration. In this case, the "Surface tension" of the liquid and the "Breaking ratio" must be specified. Optionally, you can indicate the indices of the distribution  $\sigma$  (Amplitude of the Lognormal distribution) and  $Y$  (Amplitude of the Rosin-Rammler distribution).

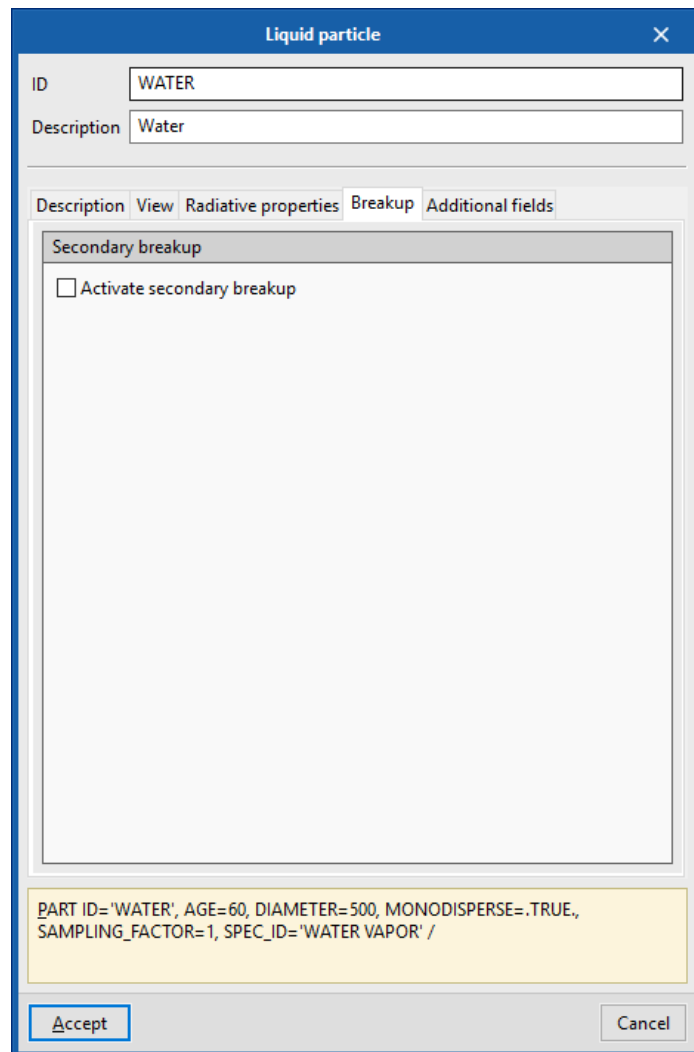


Fig. 28. Liquid particle, breakup

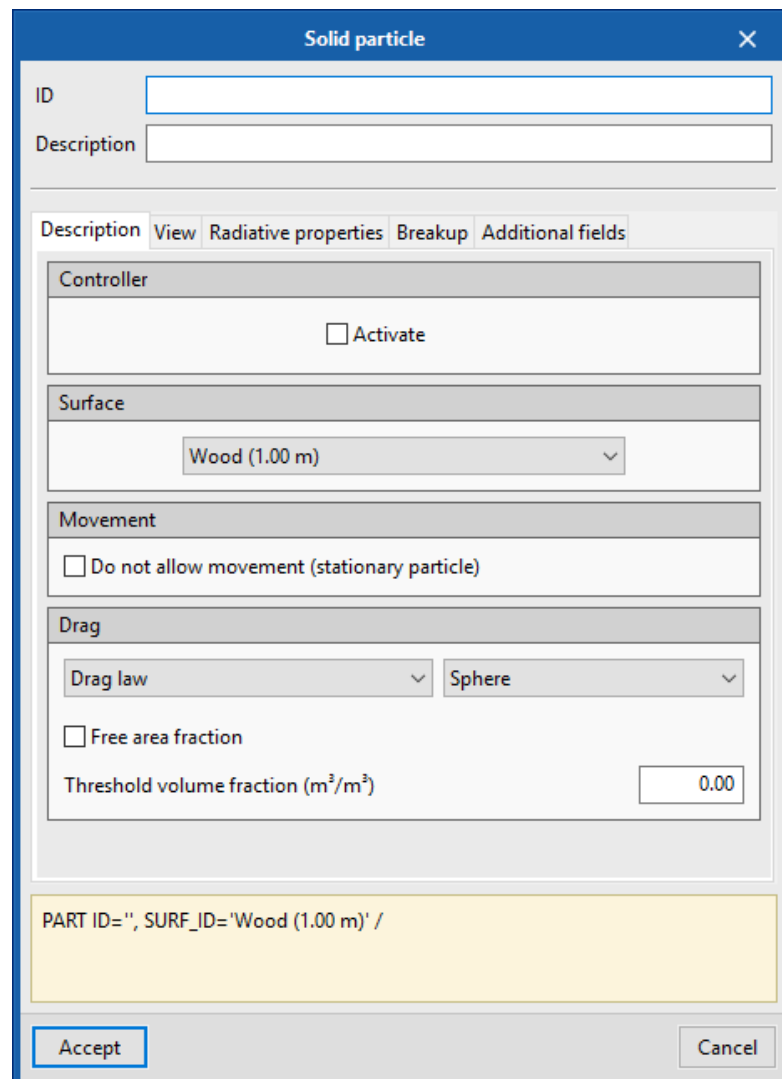
### 3.5.3 *Solid particles*

The definition of the "Visualization", the "Radiative Properties" and the "Disintegration" of the solid particles is the same as that corresponding to the liquid particles.

#### 3.5.3.1 *Description*

- **Controller.** FDS can control the presence of solid particles in the simulation depending on the response of a control or device.
- **Surface.** To define a solid particle, you must specify the surface that forms it and gives it its properties.

- **Movement.** It is possible to indicate to the FDS calculation engine if a solid particle is stationary.
- **Drag.** Indicates the correlation of the drag coefficient as a function of the Reynolds number based on the diameter of the particle.



The image shows a 'Solid particle' dialog box with the following fields and options:

- ID:** A text input field.
- Description:** A text input field.
- Controller:** A section with an ☐ **Activate** checkbox.
- Surface:** A dropdown menu currently showing 'Wood (1.00 m)'.
- Movement:** A section with an ☐ **Do not allow movement (stationary particle)** checkbox.
- Drag:** A section containing:
  - A 'Drag law' dropdown menu.
  - A 'Sphere' dropdown menu.
  - An ☐ **Free area fraction** checkbox.
  - A 'Threshold volume fraction (m<sup>3</sup>/m<sup>3</sup>)' input field with the value '0.00'.
- Summary:** A yellow box at the bottom containing the text: `PART ID="", SURF_ID='Wood (1.00 m)' /`
- Buttons:** 'Accept' and 'Cancel' buttons at the bottom.

Fig. 29. Solid particle, description

### 3.5.4 *Particles without mass*

Massless particles are the simplest Lagrangian particles and are used only for visualization.

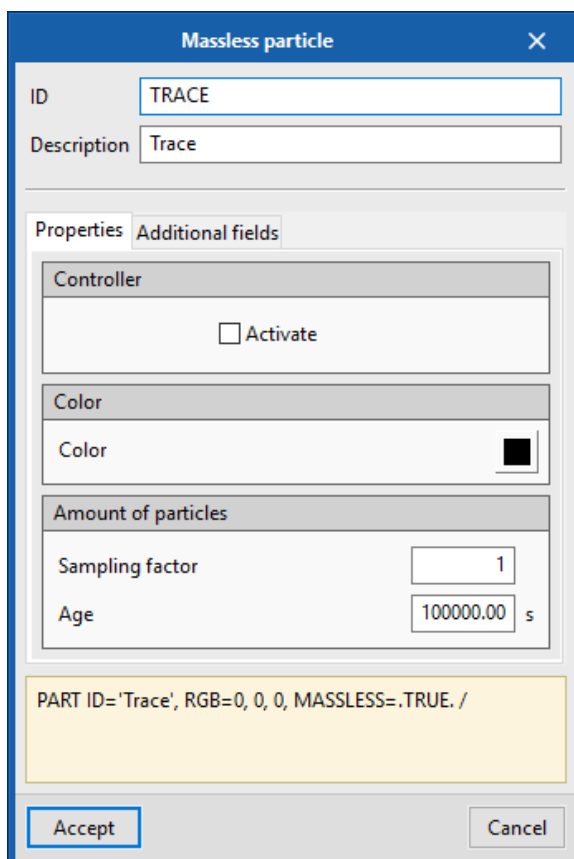


Fig. 30. Massless particle, properties

## 3.6 Reactions

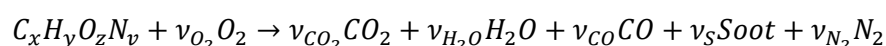
To define the combustion model that will be carried out during the simulation, the FDS calculation engine allows the specification of the parameters that govern the behaviour of the reaction.

Due to the cost of solving the transport equations, the FDS calculation engine only allows an active reaction in the model. Consequently, it must be indicated in CYPEFIRE FDS which reaction must be taken into account during the simulation using the Active option.

### 3.6.1 Fuel

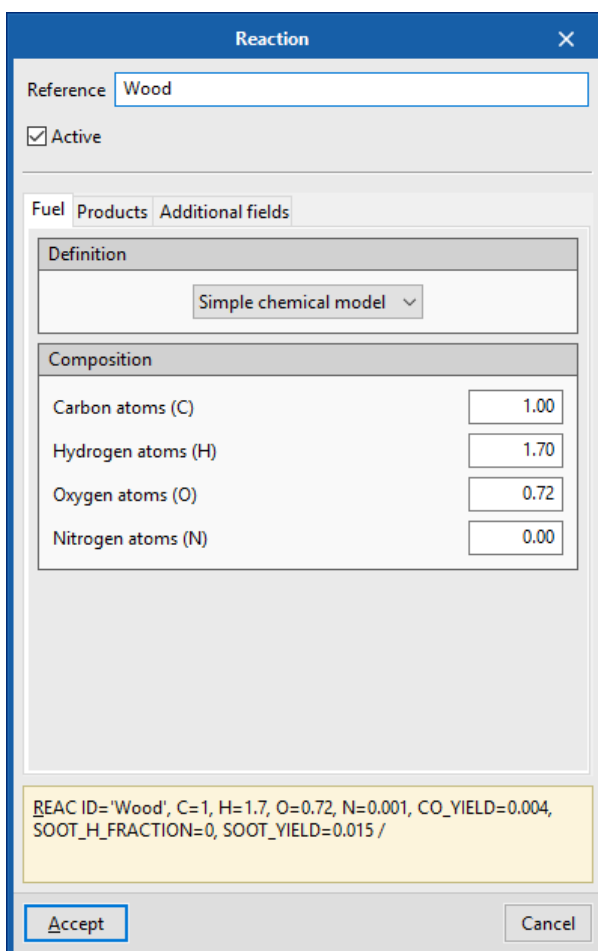
Within this tab CYPEFIRE FDS allows to select the method of definition of the type of fuel that will be used in the reaction. The two options offered by the program are the following:

- **Simple chemical model.** In the simple chemical model the form of the reaction is always assumed as:



Consequently, to define the chemical formula of the fuel, the number of carbon atoms (C), hydrogen (H), oxygen (O) and nitrogen (N) must be specified.

- **By type of species.** By defining the reaction in this way you can select a species that acts as fuel, previously defined in the project.



Reaction

Reference: Wood

☒ Active

Fuel Products Additional fields

Definition

Simple chemical model

Composition

Carbon atoms (C)	1.00
Hydrogen atoms (H)	1.70
Oxygen atoms (O)	0.72
Nitrogen atoms (N)	0.00

REAC ID= 'Wood', C=1, H=1.7, O=0.72, N=0.001, CO\_YIELD=0.004, SOOT\_H\_FRACTION=0, SOOT\_YIELD=0.015 /

Accept Cancel

Fig. 31. Reaction, fuel

### 3.6.2 *Products*

CYPEFIRE FDS allows you to specify the amount of energy emitted by the combustion reaction using the following parameters:

- **Heat combustion.** It is used in the calculation of the enthalpy of fuel formation to subsequently determine the emission of energy per unit volume of the chemical reaction.
- **EPUMO2.** Indicates the amount of energy emission per unit mass of oxygen consumed. It is used to determine the Heat of combustion when its value is not known.
- **Radiation fraction.** This option allows the explicit specification of the fraction of the total combustion energy that is emitted in the form of thermal radiation.
- **Ideal combustion heat.** When activating this option, the Heat of combustion will be reduced based on the values of Fraction of mass transformed into soot and Fraction of mass transformed into CO. If EPUMO2 is defined instead of the Heat of combustion, its value will not change.

The coefficients of the reaction products of the simple chemical model are computed based on the following parameters:

- **Fraction of the mass transformed into soot.** Indicates the fraction of fuel mass transformed into smoke particles.
- **Fraction of the mass transformed into CO.** Indicates the fraction of fuel mass transformed into carbon monoxide.
- **Fraction of hydrogen atoms in soot.**

From this data, the calculation engine FDS calculates the coefficients automatically in the following way:

$$v_{O_2} = v_{CO_2} + \frac{v_{CO}}{2} + \frac{v_{H_2O}}{2} - \frac{Z}{2}$$

$$v_{CO_2} = x - v_{CO} - (1 - X_H)v_S$$

$$v_{H_2O} = \frac{y}{2} - \frac{X_H}{2}v_S$$

$$v_{CO} = \frac{W_F}{W_{CO}}y_{CO}$$

$$v_S = \frac{W_F}{W_S}y_S$$

$$v_{N_2} = \frac{v}{2}$$

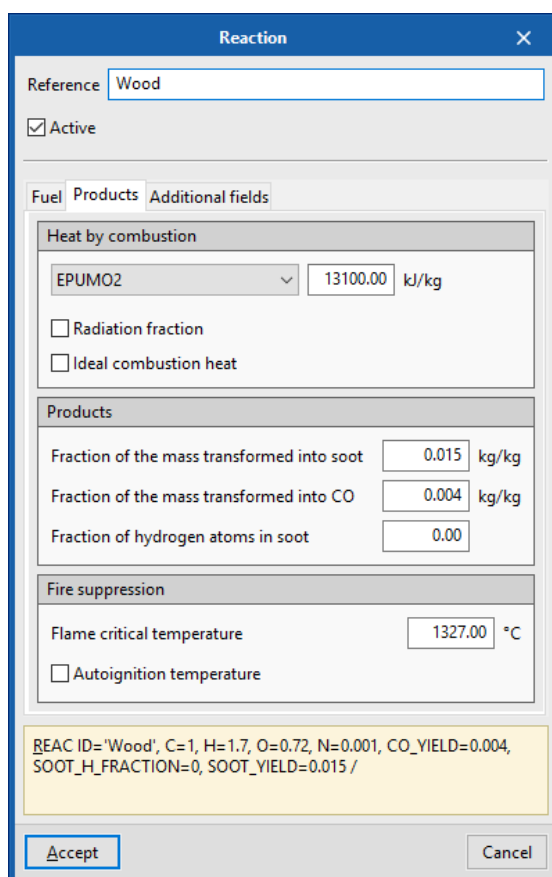
$$W_S = X_H W_H + (1 - X_H) W_C$$

Where:

- $y_S$ : Mass fraction transformed into soot
- $y_{CO}$ : Fraction of mass transformed into CO
- $X_H$ : Fraction of hydrogen atoms in soot

Finally, it is possible to specify the values of the parameters that govern the suppression of fire, these are:

- **Flame critical temperature.** Indicates the temperature at which a diffusion flame ceases to exist. In the combustion process, a diffusion flame is one in which the oxidant is combined with the fuel by diffusion.
- **Auto ignition temperature.** Indicates the minimum temperature for combustion to occur.



Reaction

Reference: Wood

☒ Active

Fuel Products Additional fields

Heat by combustion

EPUMO2 13100.00 kJ/kg

☐ Radiation fraction

☐ Ideal combustion heat

Products

Fraction of the mass transformed into soot 0.015 kg/kg

Fraction of the mass transformed into CO 0.004 kg/kg

Fraction of hydrogen atoms in soot 0.00

Fire suppression

Flame critical temperature 1327.00 °C

☐ Autoignition temperature

REAC ID='Wood', C=1, H=1.7, O=0.72, N=0.001, CO\_YIELD=0.004, SOOT\_H\_FRACTION=0, SOOT\_YIELD=0.015 /

Accept Cancel

Fig. 32. Reaction, products



## 3.7 Device models

### 3.7.1 *Heat detector models*

Heat detectors are devices capable of measuring temperature at a point using the activation algorithm based on the Response Time Index, as a sprinkler but without emission.

$$\frac{dT_l}{dt} = \frac{\sqrt{|u|}}{RTI} (T_g - T_l) - \frac{C}{RTI} (T_l - T_m) - \frac{C_2}{RTI} \beta |u|$$

Where:

$u$ : Gas velocity

$RTI$ : Response time index

$T_l$ : Sensor temperature

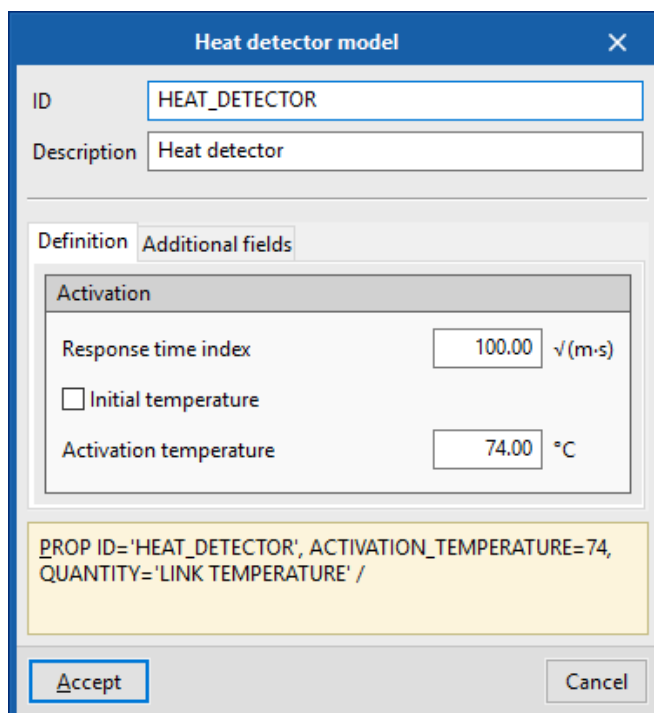
$T_g$ : Gas temperature close to the sensor

$T_m$ : Support temperature

$\beta$ : Fraction of volume of liquid water in the gas

$C$ : Experimentally determined factor that indicates the amount of heat conducted out of the sensor by the support

$C_2$ : Constant determined empirically by DiMarzo ( $6 \times 10^6 \text{ K}/(\text{m/s})^{1/2}$ )



The dialog box titled "Heat detector model" contains the following fields and options:

- ID:** HEAT\_DETECTOR
- Description:** Heat detector
- Definition:**
  - Activation:**
    - Response time index:** 100.00  $\sqrt{(\text{m}\cdot\text{s})}$
    - ☐ Initial temperature
    - Activation temperature:** 74.00  $^{\circ}\text{C}$
- Summary:** PROP ID='HEAT\_DETECTOR', ACTIVATION\_TEMPERATURE=74, QUANTITY='LINK TEMPERATURE' /
- Buttons:** Accept, Cancel

Fig. 33. Heat detector model

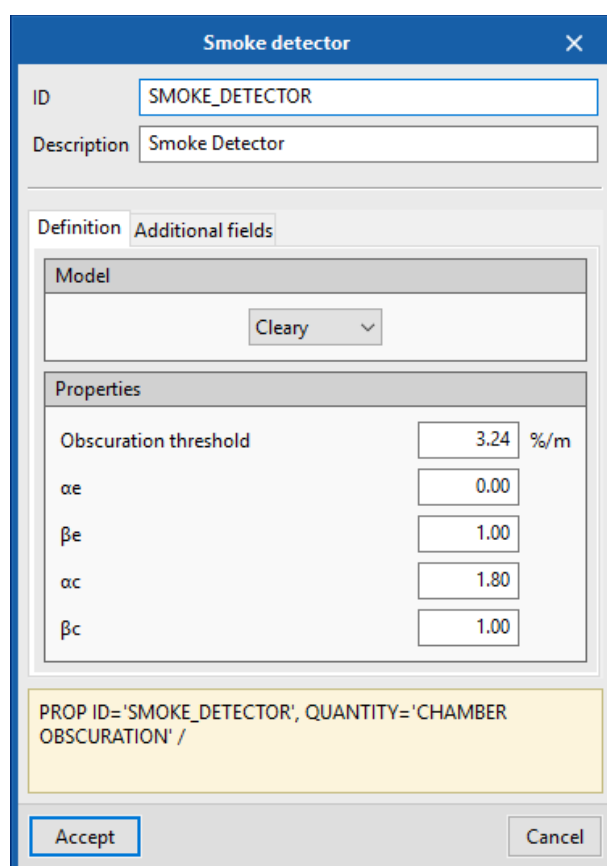
### 3.7.2 *Models of smoke detectors*

Smoke detectors are devices capable of measuring the level of obscuration at a point. The FDS calculation engine allows the introduction of Heskestad and Cleary smoke detector models.

- **Heskestad.** The mass fraction of smoke in the detection chamber of the detector  $Y_c$  is located behind the mass fraction in the external free flow  $Y_e$  in a period of time  $\delta t = L / u$ , where  $u$  is the velocity of the free flow and  $L$  is the characteristic length of the detector geometry. The change in the mass fraction of the smoke in the detection chamber can be found by solving the following equation:

$$\frac{dY_c}{dt} = \frac{Y_e(t) - Y_c(t)}{L/u}$$

The detector is activated when  $Y_c$  exceeds the detector darkening threshold.



The dialog box titled "Smoke detector" contains the following fields and sections:

- ID:** SMOKE\_DETECTOR
- Description:** Smoke Detector
- Definition / Additional fields:**
  - Model:** Cleary (dropdown menu)
  - Properties:**

Obscuration threshold	3.24 %/m
$\alpha_e$	0.00
$\beta_e$	1.00
$\alpha_c$	1.80
$\beta_c$	1.00
- Summary:** PROP ID= 'SMOKE\_DETECTOR', QUANTITY= 'CHAMBER OBSCURATION' /
- Buttons:** Accept, Cancel

Fig. 34. Smoke detector

- **Cleary.** The Cleary model involves two filling times instead of one. The smoke must first pass through the outer casing and then through a series of baffles before reaching the detection chamber. There is a time lag associated with the passage of smoke through the housing and also with the entry of smoke into the detection chamber. The parameter  $\delta t_e$  indicates the characteristic filling time of the entire volume enclosed in the external housing. The parameter  $\delta t_c$  indicates the characteristic filling time of the detection chamber. Cleary suggested that each characteristic fill time is a function of the free flow velocity  $u$  outside the detector.

$$\delta t_e = \alpha_e u^{\beta_e} \quad ; \quad \delta t_c = \alpha_c u^{\beta_c}$$

The parameters  $\alpha$  and  $\beta$  are empirical constants related to the specific geometry of the detector. The change in the mass fraction of smoke in the detection chamber  $Y_c$  can be determined by solving the following equation:

$$\frac{dY_c}{dt} = \frac{Y_e(t - \delta t_e) - Y_c(t)}{\delta t_c}$$

Where  $Y_e$  is the mass fraction of the smoke outside the detector in the free flow.

### 3.7.3 *Thermocouple models*

The output value of a thermocouple is the temperature of the device itself. The temperature of the thermocouple differs from the gas by an amount relative to the size of the sphere. This can be determined using the thermocouple temperature equation,  $T_{TC}$ .

$$\rho_{TC} c_{TC} \frac{dT_{TC}}{dt} = \varepsilon_{TC} (U/4 - \sigma T_{TC}^4) + h(T_g - T_{TC}) = 0$$

Where:

$\varepsilon_{TC}$ : Emissivity of the thermocouple

$U$ : Integrated radiative intensity

$T_g$ : Actual gas temperature

$h$ : Coefficient of heat transmission to a small sphere,  $h = kNu/D_{TC}$

In the thermocouple model panel you must specify the **diameter** value, the **emissivity**, the **density** and the **specific heat** of the sphere. It is possible to force the value of the **heat transfer coefficient** in case you do not want to use the calculated value. The default values for the density and specific heat of the sphere are those of nickel; 8908 kg / m<sup>3</sup> and 0.44 kJ / kg / K.

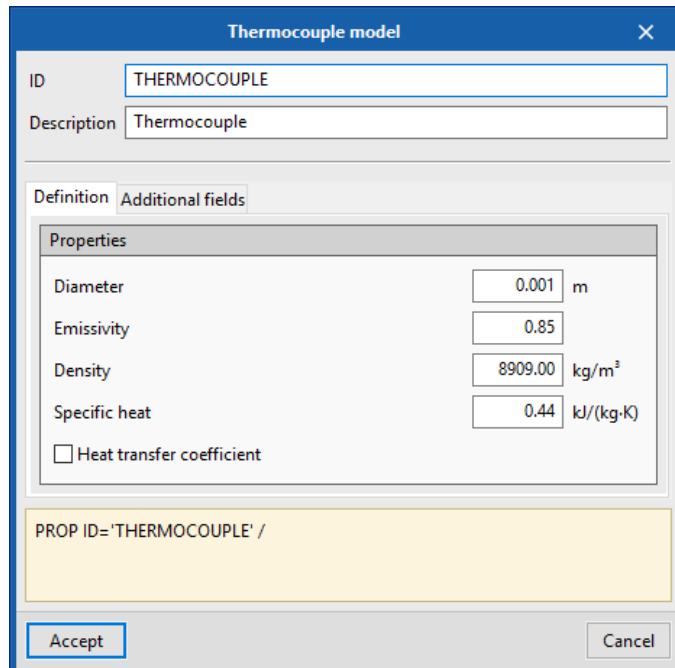


Fig. 35. Thermocouple model

### 3.7.4 *Sprinkler models*

#### 3.7.4.1 *Definition*

- **Particle.** Indicates the particle that is going to be sprayed by the device.
- **Activation.** In the FDS calculation engine, sprinklers are activated according to the standard RTI algorithm (Response Time Index).

$$2. \quad \frac{dT_l}{dt} = \frac{\sqrt{|u|}}{RTI} (T_g - T_l) - \frac{C}{RTI} (T_l - T_m) - \frac{C_2}{RTI} \beta |u|$$

Where:

$u$ : Gas velocity

$RTI$ : Response time index

$T_l$ : Sensor temperature

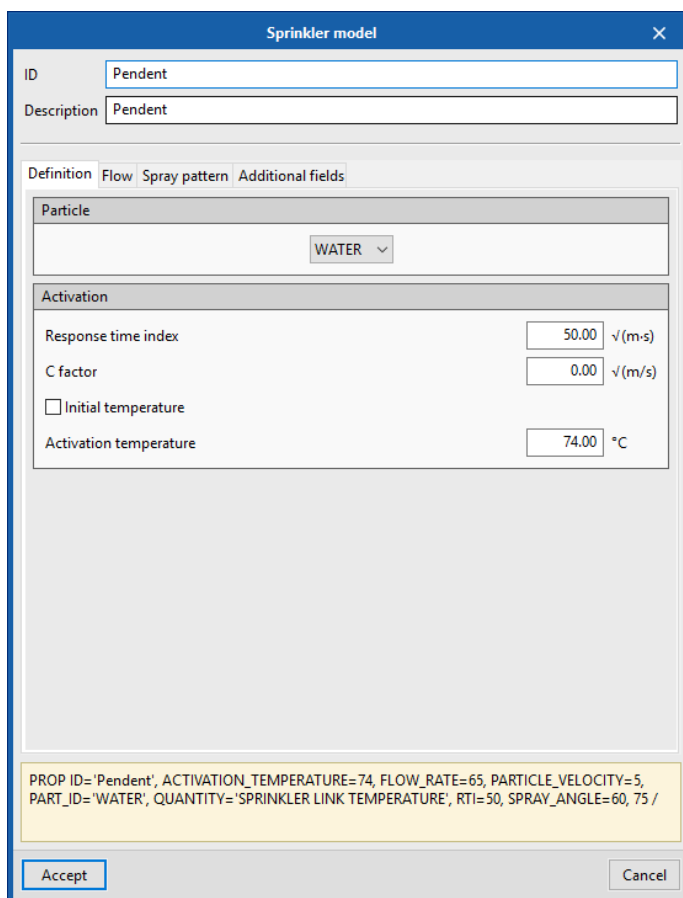
$T_g$ : Gas temperature close to the sensor

$T_m$ : Sprinkler support temperature

$\beta$ : Fraction of volume of liquid water in the gas

$C$ : Experimentally determined factor that indicates the amount of heat conducted out of the sensor by the support

$C_2$ : Constant determined empirically by DiMarzo ( $6 \times 10^6 \text{ K}/(\text{m/s})^{1/2}$ )



The dialog box titled "Sprinkler model" contains the following fields and tabs:

- ID:** Pendent
- Description:** Pendent
- Tabs:** Definition (selected), Flow, Spray pattern, Additional fields
- Particle:** WATER (dropdown menu)
- Activation:**
  - Response time index: 50.00 (m-s)
  - C factor: 0.00 (m/s)
  - ☐ Initial temperature
  - Activation temperature: 74.00 °C
- Status Bar:** PROP ID='Pendent', ACTIVATION\_TEMPERATURE=74, FLOW\_RATE=65, PARTICLE\_VELOCITY=5, PART\_ID='WATER', QUANTITY='SPRINKLER LINK TEMPERATURE', RTI=50, SPRAY\_ANGLE=60, 75 /
- Buttons:** Accept, Cancel

Fig. 36. Sprinkler model, definition

### 3.7.4.2 *Flow*

- **Flow.** The spray flow can be indicated by specifying the value of the "Volumetric flow" or the "Mass flow rate". As an alternative, it is possible to indicate the value of the "K-Factor" and the "Operating Pressure". In this last case the volumetric flow is calculated from the following equation:

$$K\sqrt{p}$$

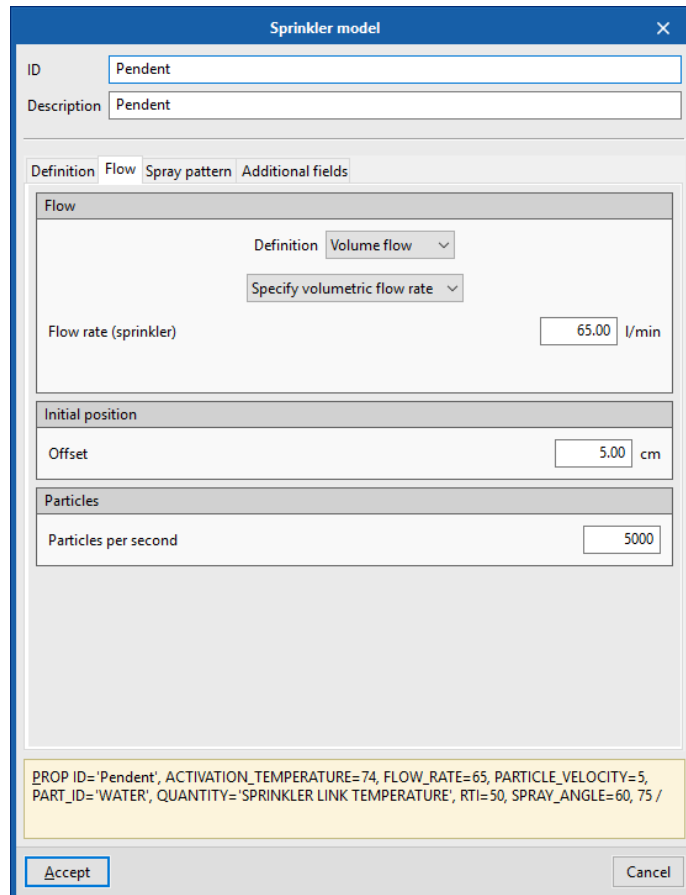
Where:

$K$ : Factor K [L/(min\*bar<sup>1/2</sup>)]

$p$ : Operating pressure [bar]

- **Initial position.** The value of the radius (m) of a sphere around the sprinkler where the water droplets are initially placed in the simulation is indicated. It is assumed that beyond this sphere the drops have completely broken and move independently.

- **Particles.** The number of particles inserted every second is indicated by each active sprinkler.



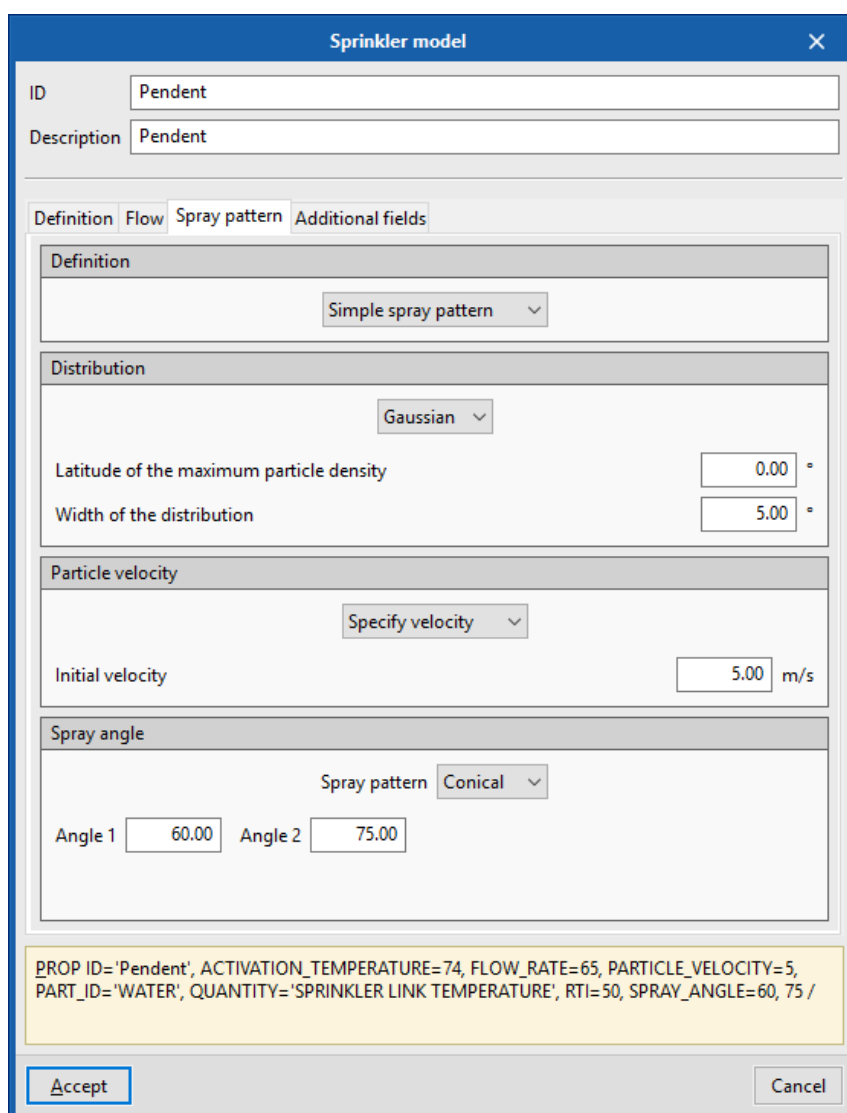
The image shows a software dialog box titled "Sprinkler model". It contains several input fields and tabs. The "Definition" tab is selected, showing fields for "ID" (Pendent), "Description" (Pendent), "Flow rate (sprinkler)" (65.00 l/min), "Initial position" (Offset: 5.00 cm), and "Particles" (Particles per second: 5000). The "Flow" tab is also visible, showing "Definition" (Volume flow) and "Specify volumetric flow rate". At the bottom, there is a yellow box with a summary of the configuration: "PROP ID='Pendent', ACTIVATION\_TEMPERATURE=74, FLOW\_RATE=65, PARTICLE\_VELOCITY=5, PART\_ID='WATER', QUANTITY='SPRINKLER LINK TEMPERATURE', RTI=50, SPRAY\_ANGLE=60, 75 /". The dialog has "Accept" and "Cancel" buttons at the bottom.

Fig. 37. Sprinkler model, flow

### 3.7.4.3 *Sprinkler pattern*

- **Simple pattern.**
  - **Distribution.** Indicates how the drops are distributed within the specified "Spray Angle". The available options are "Gaussian" and "Uniform". In the case of selecting the "Gaussian" distribution, you must specify the value of the "Latitude of the maximum density of particles" and the "Width of the distribution".
  - **Speed of the particles.** Indicates the speed of the drops at their insertion point.
  - **Spray angle.** The pair of angles through which the particles are sprayed is indicated. The angles draw a conical pattern relative to the south pole of the sphere centered on the sprinkler with a radius equal to that indicated for the "Initial position".
- **Complex pattern.**

- **Table.** For complex spray patterns, it is possible to use a table of values with the spherical distribution for each solid angle. "Latitude 1" and "Latitude 2" are the limits of the solid angle measured in degrees from the south pole (0 is the south pole and 90 is the equator, 180 is the north pole). The parameters "Length 1" and "Length 2" are the limits of the solid angle (also in degrees), where 0 (or 360) is aligned with the x axis and 90 is aligned with the y axis. Finally, you must specify the "Velocity" of the drops at their insertion point and the fraction of the total liquid flow that must emerge from that particular solid angle ("Flow fraction").



**Sprinkler model**

ID: Pendent  
Description: Pendent

Definition | Flow | **Spray pattern** | Additional fields

**Definition**  
Simple spray pattern

**Distribution**  
Gaussian  
Latitude of the maximum particle density: 0.00 °  
Width of the distribution: 5.00 °

**Particle velocity**  
Specify velocity  
Initial velocity: 5.00 m/s

**Spray angle**  
Spray pattern: Conical  
Angle 1: 60.00 Angle 2: 75.00

PROP ID='Pendent', ACTIVATION\_TEMPERATURE=74, FLOW\_RATE=65, PARTICLE\_VELOCITY=5, PART\_ID='WATER', QUANTITY='SPRINKLER LINK TEMPERATURE', RTI=50, SPRAY\_ANGLE=60, 75 /

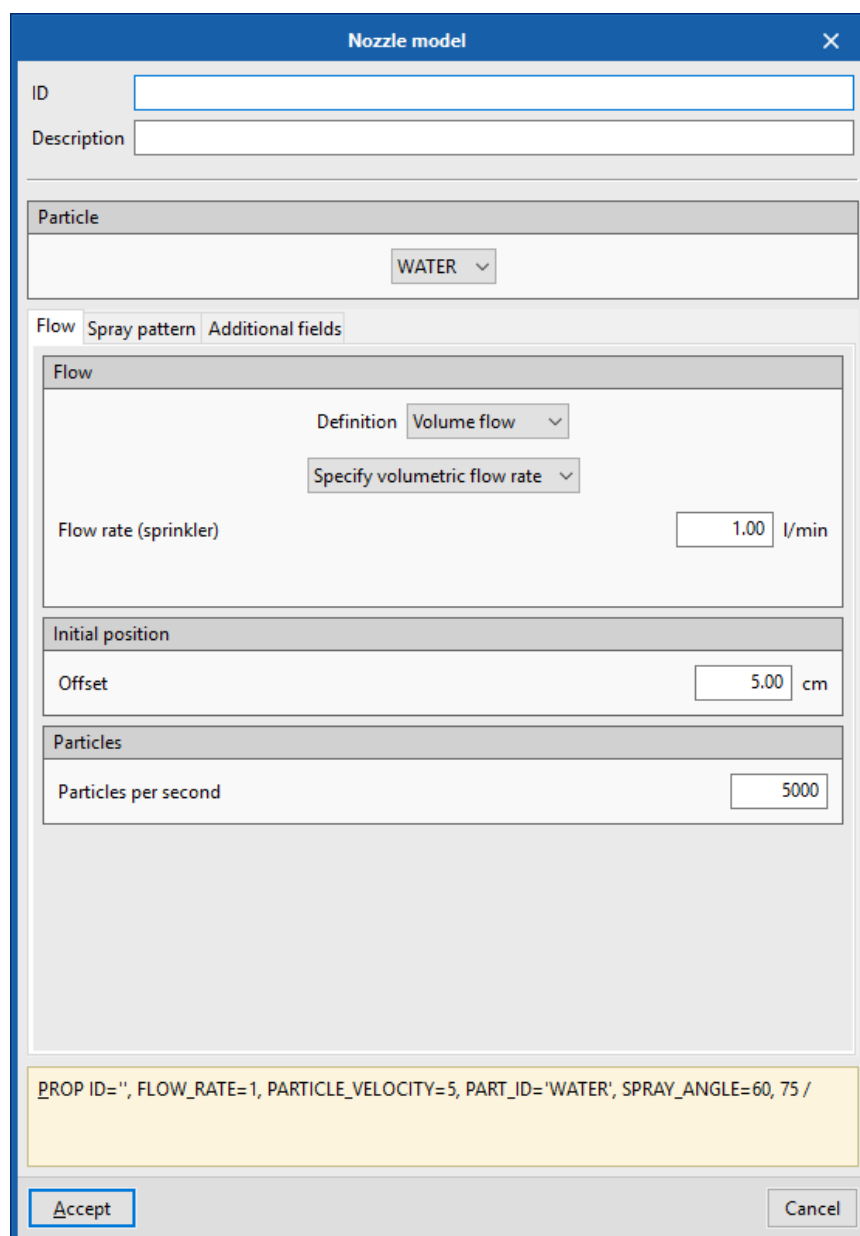
Accept Cancel

Fig. 38. Sprinkler model, spray pattern

### 3.7.5 *Nozzle models*

The nozzles are devices similar to sprinklers, the difference is that they are not activated based on the standard model RTI (Response Time Index).

The definition of "Flow" and "Sprinkler pattern" is equivalent to that described in the previous section corresponding to the "Sprinkler models".



**Nozzle model**

ID:

Description:

Particle:

Flow | Spray pattern | Additional fields

**Flow**

Definition:

Specify volumetric flow rate:  l/min

Flow rate (sprinkler):  l/min

**Initial position**

Offset:  cm

**Particles**

Particles per second:

PROP ID=", FLOW\_RATE=1, PARTICLE\_VELOCITY=5, PART\_ID='WATER', SPRAY\_ANGLE=60, 75 /

Fig. 39. Nozzle model, flow



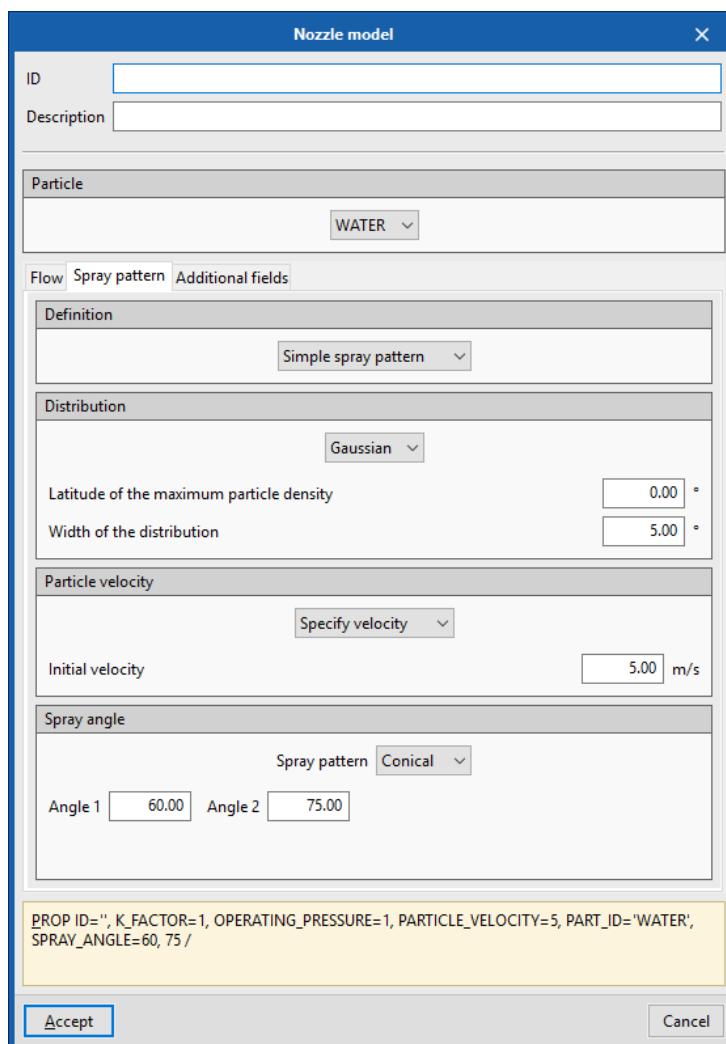


Fig. 40. Nozzle model, spray pattern

## 3.8 Meshes

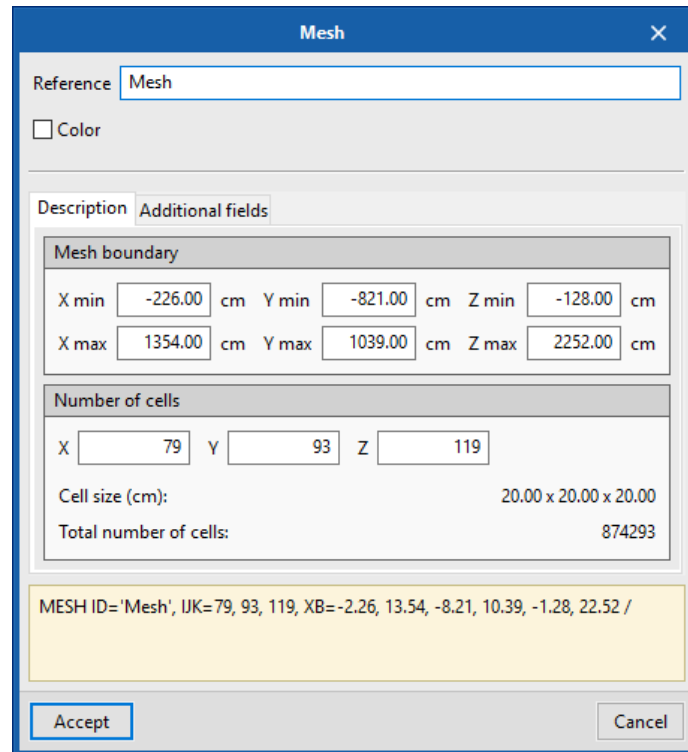
The meshes constitute the computational region of the simulation, which implies that all the objects involved in the calculation must be inside a mesh. A mesh is a parallelepiped defined by two points that correspond to two of its opposite corners.

In a simulation, more than one mesh can be introduced in order to execute the calculation in parallel through MPI (Message Passing Interface).

Each mesh is subdivided into uniform cells. The size of the cells should be approximately the same in all three directions to obtain an optimum degree of accuracy.

It should be noted that the FDS calculation engine uses the fast Fourier transform (FFTs) to determine the pressure in the y and z directions, and this algorithm is more efficient when the number of cells in those directions can be factored into low cousins, such as 2, 3 and 5.

The number of cells in the x direction is not affected because FFTs are not used in the x direction.



The image shows a 'Mesh' dialog box with the following fields and values:

- Reference: Mesh
- Color: ☐
- Description: Additional fields
- Mesh boundary:
  - X min: -226.00 cm
  - Y min: -821.00 cm
  - Z min: -128.00 cm
  - X max: 1354.00 cm
  - Y max: 1039.00 cm
  - Z max: 2252.00 cm
- Number of cells:
  - X: 79
  - Y: 93
  - Z: 119
  - Cell size (cm): 20.00 x 20.00 x 20.00
  - Total number of cells: 874293
- MESH ID= 'Mesh', IJK=79, 93, 119, XB=-2.26, 13.54, -8.21, 10.39, -1.28, 22.52 /
- Buttons: Accept, Cancel

Fig. 41. Mesh

## 3.9 Zones

Through the pressure zones it is possible to model closed regions of the computational domain that have their own level of ambient pressure. The FDS calculation engine is not able to identify these spaces based solely on the location of the obstructions, therefore, it is necessary to indicate where they are located and what their dimensions are. To do this, you must specify in CYPEFIRE FDS the values of the geometric coordinates that define the edges of the rectangular area. In case the region is not rectangular, the FDS calculation engine will extend the boundaries of the zone to make up the non-rectangular region.

To indicate that a pressure zone is linked to another, through ducts or other connections, CYPEFIRE FDS allows to indicate the existence of a leakage area and the dimensions of it. By default, the *External* zone is defined, which has the ambient pressure introduced in the *Environment* options of the toolbar.

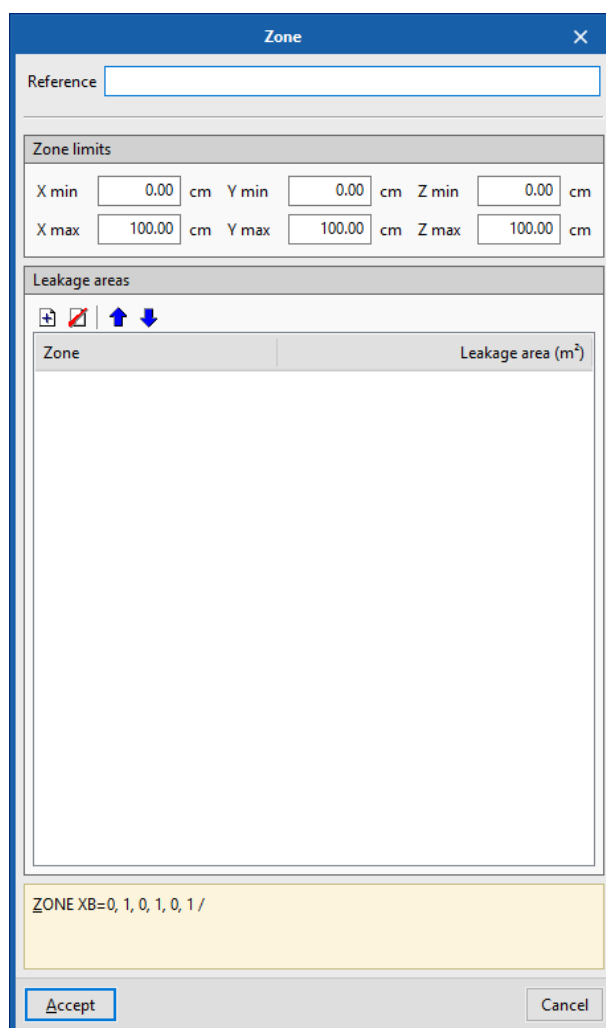
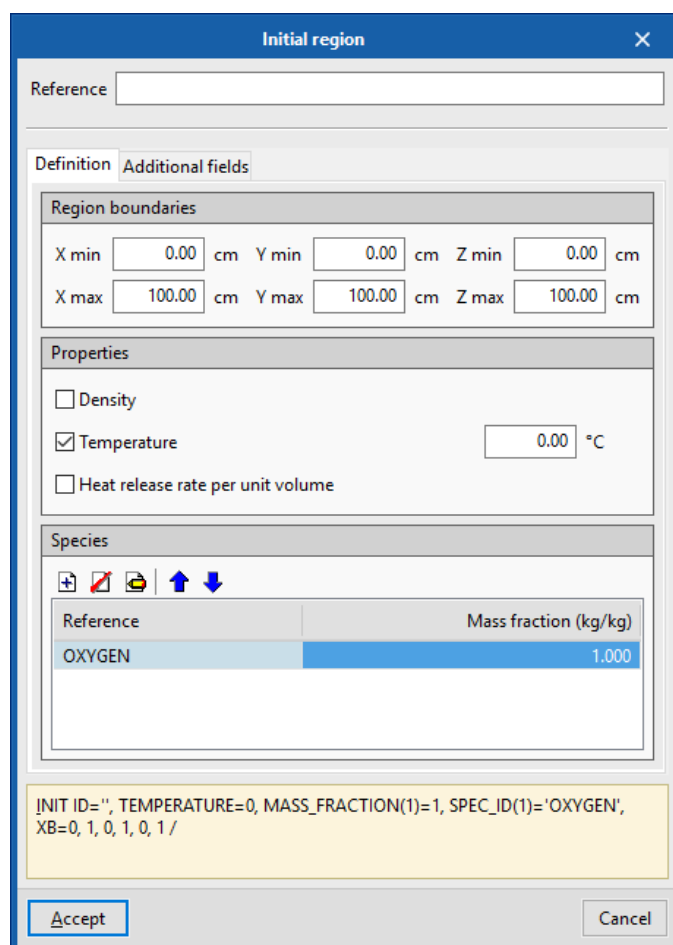


Fig. 42. Zone

### 3.9.1 *Initial regions*

An FDS simulation begins with the environmental conditions defined in the "Environment" options for the entire scene. This is usually valid for most cases; however, it is sometimes convenient to modify the environmental conditions in a specific region of the computational domain. The initial regions are rectangular spaces that allow this function to be carried out.

- **Limits of the region.** Indicates the coordinates of the two points that define the position and dimensions of the initial region.
- **Properties.** It is possible to indicate a value of "Density", "Temperature" and "Emission of heat per unit volume" different from that defined in the "Environment" options.
- **Species.** Indicates the presence and concentration of species in the initial region.



The dialog box is titled "Initial region" and has a close button (X) in the top right corner. It contains several sections:

- Reference:** A text input field.
- Definition / Additional fields:** A tabbed interface with "Definition" selected.
- Region boundaries:** A section with input fields for X min (0.00 cm), Y min (0.00 cm), Z min (0.00 cm), X max (100.00 cm), Y max (100.00 cm), and Z max (100.00 cm).
- Properties:** A section with checkboxes for "Density", "Temperature" (checked), and "Heat release rate per unit volume". The "Temperature" field is set to 0.00 °C.
- Species:** A section with a table showing the mass fraction of species. The table has two columns: "Reference" and "Mass fraction (kg/kg)". The first row is "OXYGEN" with a mass fraction of 1.000.
- Code:** A text area at the bottom containing the following code:
 

```
[INIT ID=", TEMPERATURE=0, MASS_FRACTION(1)=1, SPEC_ID(1)='OXYGEN',
XB=0, 1, 0, 1, 0, 1 /
```
- Buttons:** "Accept" and "Cancel" buttons at the bottom.

Fig. 43. Initial region

## 3.10 Obstructions

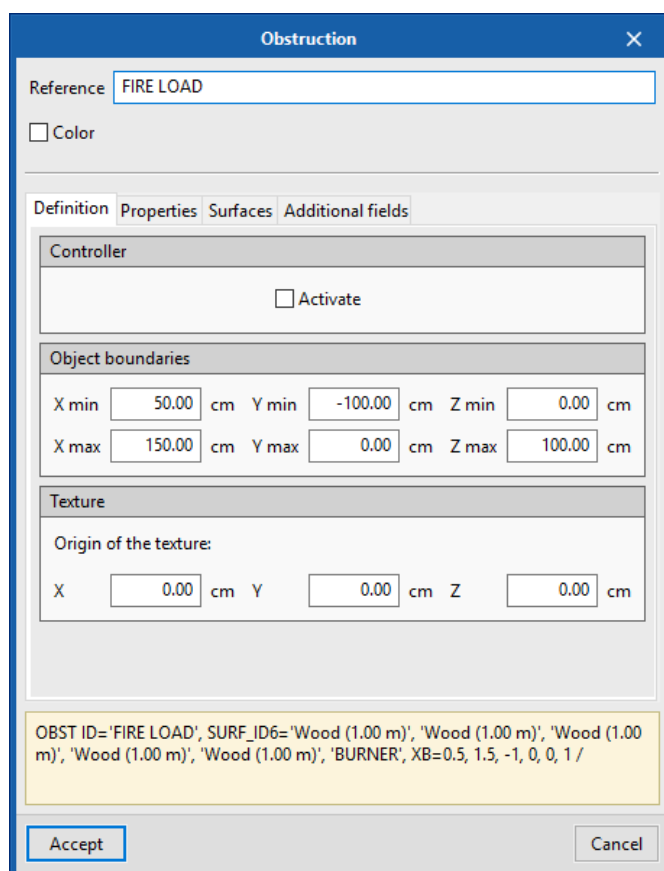
The obstructions are rectangular solids whose sides are parallel to the planes of the coordinate axis. These are the main FDS entities used to generate the 3D geometric model in order for it to be simulated.

The colour of the complete obstructions can be indicated, and in the case that they are not, the colours on the surfaces of each face of the object will be used in the simulation instead.

### 3.10.1 *Definition*

General parameters can be defined in this tab such as the definition of the project's geometry and the activation of the obstruction.

- **Controller.** The window shows how the FDS controls the presence of an obstruction in the simulation depending on the response of a control or device. This way, events can be modelled such as the breakage of the windows due to the rise of the temperature.
- **Limits of the object.** This window indicates the coordinates of the two points that define the position and dimensions of the obstruction.
- **Texture.** This window indicates the origin of the texture used in the surfaces that make up the obstruction.



The image shows a software window titled "Obstruction" with a close button (X) in the top right corner. The window contains several sections:

- Reference:** A text box containing "FIRE LOAD".
- Color:** A checkbox labeled "Color" which is currently unchecked.
- Definition:** A tabbed interface with three tabs: "Definition" (selected), "Properties", and "Surfaces".
- Controller:** A section with an "Activate" checkbox, which is unchecked.
- Object boundaries:** A section with input fields for coordinates in centimeters:
 

X min	50.00	cm	Y min	-100.00	cm	Z min	0.00	cm
X max	150.00	cm	Y max	0.00	cm	Z max	100.00	cm
- Texture:** A section with the label "Origin of the texture:" and input fields for X, Y, and Z coordinates in centimeters:
 

X	0.00	cm	Y	0.00	cm	Z	0.00	cm
---	------	----	---	------	----	---	------	----
- Preview:** A yellow highlighted box at the bottom containing the text:
 

```
OBST ID='FIRE LOAD', SURF_ID6='Wood (1.00 m)', 'Wood (1.00 m)', 'Wood (1.00 m)', 'Wood (1.00 m)', 'Wood (1.00 m)', 'BURNER', XB=0.5, 1.5, -1, 0, 0, 1 /
```
- Buttons:** "Accept" and "Cancel" buttons at the bottom.

Fig. 44. Obstruction, definition

### 3.10.2 *Properties*

In this tab, the main properties of the obstruction can be accessed and the visualization and interpretation of its geometry can be generated by the FDS engine.

- **Allow gaps (HOLE).** Activating this option will allow the obstruction to be penetrated by hole-type objects (HOLE).

- **Allow vents (VENT).** Activating this option will allow the placement of vent-type objects (VENT) on the surfaces of the obstruction.
- **Rejectable.** Activating this option will allow the program to discard the obstruction of the calculation.
- **Overlap.** If the faces of two obstructions overlap, FDS will use the properties of the obstruction that was defined second. If this is not desired, activate this option in the second obstruction so that FDS uses the properties of the first one.
- **With thickness.** Activating this option will force the object to occupy the space of a cell in the mesh. This prevents FDS from approaching thin obstructions to 2D surfaces.
- **Draw outline.** When activating this option, the obstruction will no longer be represented as a solid in the simulation and only its outline will be shown.
- **Apparent density.** The apparent density is used to determine the mass of fuel of the solid object. This parameter overwrites the rest of the variables from which the mass of fuel would be calculated.

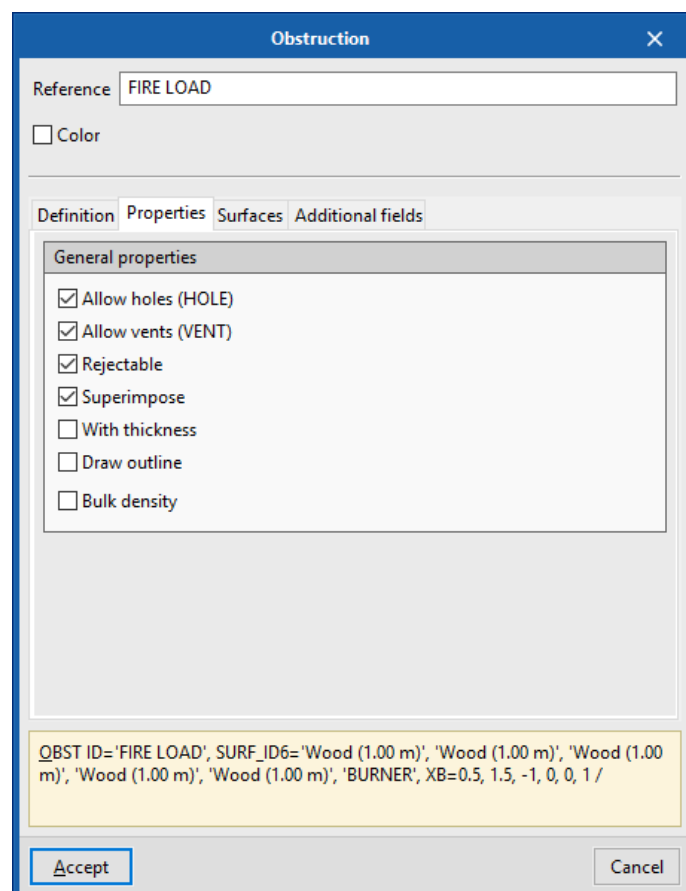
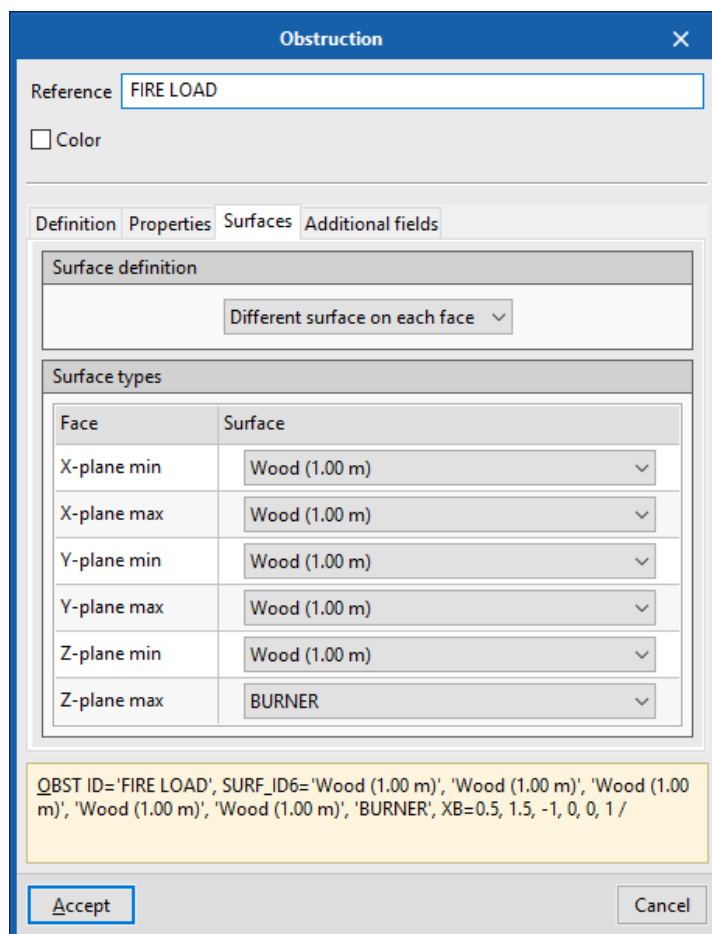


Fig. 45. Obstruction, properties

### 3.10.3 Surfaces

In this tab, types of surfaces can be indicated and generate the 6 faces of the rectangular block. Therefore, it is possible to specify a single type of surface for all the obstruction or different surfaces for each plane. By default, obstructions in FDS are created with the predefined surface INERT.



The image shows the 'Obstruction' dialog box with the 'Surfaces' tab selected. The 'Reference' field is set to 'FIRE LOAD'. The 'Color' checkbox is unchecked. The 'Definition' tab is active, showing a 'Surface definition' section with a dropdown menu set to 'Different surface on each face'. Below this is a 'Surface types' section with a table of faces and their corresponding surface types.

Face	Surface
X-plane min	Wood (1.00 m)
X-plane max	Wood (1.00 m)
Y-plane min	Wood (1.00 m)
Y-plane max	Wood (1.00 m)
Z-plane min	Wood (1.00 m)
Z-plane max	BURNER

At the bottom, a text box displays the generated FDS command: `OBST ID='FIRE LOAD', SURF_ID6='Wood (1.00 m)', 'Wood (1.00 m)', 'Wood (1.00 m)', 'Wood (1.00 m)', 'Wood (1.00 m)', 'BURNER', XB=0.5, 1.5, -1, 0, 0, 1 /`. The 'Accept' and 'Cancel' buttons are at the bottom right.

Fig. 46. Obstruction, surfaces

## 3.11 Holes

Holes are blocks that are rectangular solids whose sides are parallel to the planes of the coordinate axis, the same as the obstructions. These subtract their volume to the obstructions that intersect, whenever it has been indicated previously that they are allowed to contain holes.

By default, the holes are transparent elements in the FDS model; however, it is possible to set it to a specific colour.

### 3.11.1 *Definition*

Within this tab, the general parameters that define the geometry and activate the holes can be located.

- **Controller.** FDS controls the presence of a hole in the simulation depending on the response of the control or device.
- **Limits of the object.** Indicates the coordinates of the two points that define the position and dimensions of the hole.

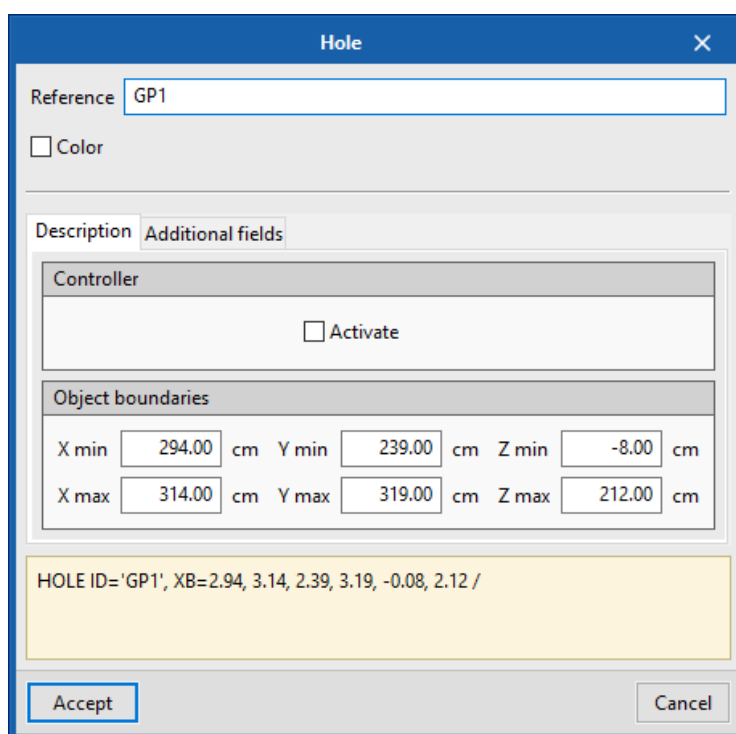


Fig. 47. Hole

## 3.12 Vents

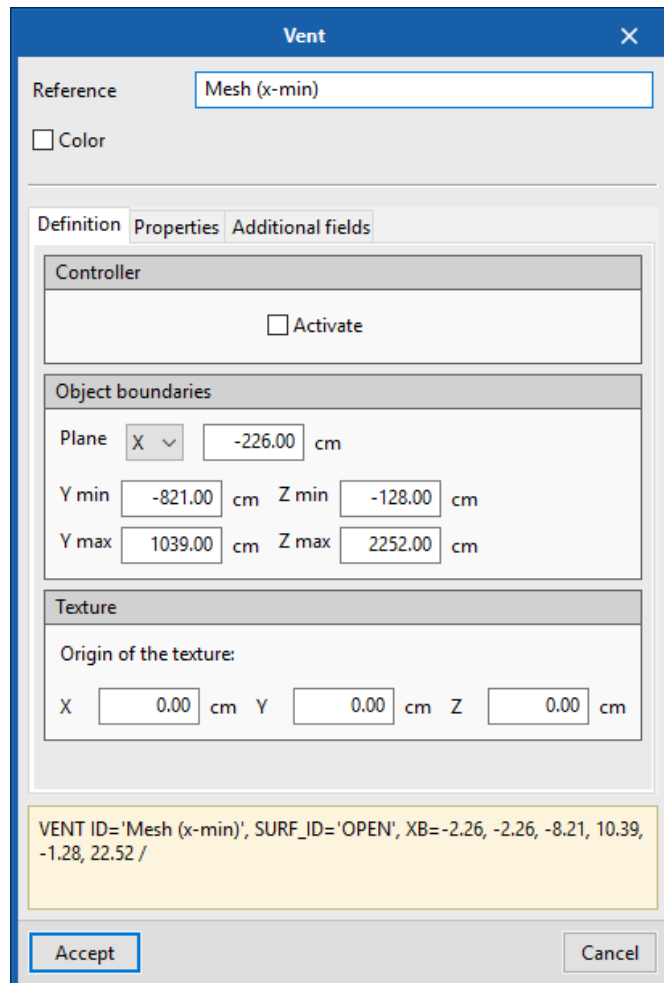
The vents are flat and adjacent to the obstructions or the boundaries of the mesh. They can be used to model components of the building's ventilation system, such as a diffuser or a return.

### 3.12.1 *Definition*

Within this tab, the general parameters that define the location and activation of the vent can be located.



- **Controller.** FDS controls the presence of a vent in the simulation depending on the response of a control or device.
- **Limits of the object.** Indicates the coordinates of the two points that define the position and dimensions of the vent.
- **Texture.** Indicates the origin of the texture used on the surface used by the vent.



**Vent**

Reference: Mesh (x-min)

☐ Color

Definition Properties Additional fields

**Controller**

☐ Activate

**Object boundaries**

Plane: X -226.00 cm

Y min: -821.00 cm Z min: -128.00 cm

Y max: 1039.00 cm Z max: 2252.00 cm

**Texture**

Origin of the texture:

X: 0.00 cm Y: 0.00 cm Z: 0.00 cm

VENT ID='Mesh (x-min)', SURF\_ID='OPEN', XB=-2.26, -2.26, -8.21, 10.39, -1.28, 22.52 /

Accept Cancel

Fig. 48. Vent, definition

### 3.12.2 Properties

Within this group of parameters are the characteristics that define the general behaviour of the vent.

- **Surface.** It is possible to indicate a surface for the vent different from the obstruction that accompanies it. The predefined surfaces OPEN, MIRROR and PERIODIC can be applied to the vents to indicate a special behaviour.

- **Geometry.** The shape of the vent can be chosen to be circular or semicircular, which means that the radius must be indicated. The normal direction of the vent can also be forced into one of the coordinate axes (positive or negative).
- **Radial propagation of fire.** It is possible to indicate to the FDS that the fire will propagate radially on the surface of the vent. To do this, the propagation speed and the starting point must be specified.

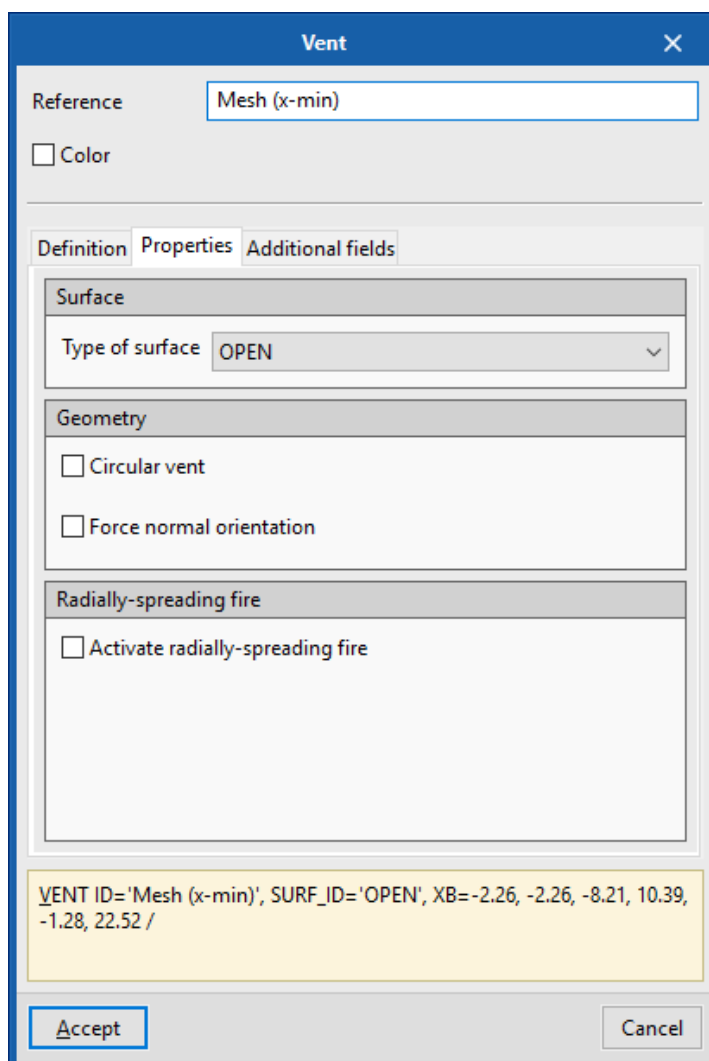


Fig. 49. Vent, properties

### 3.13 Dispositive

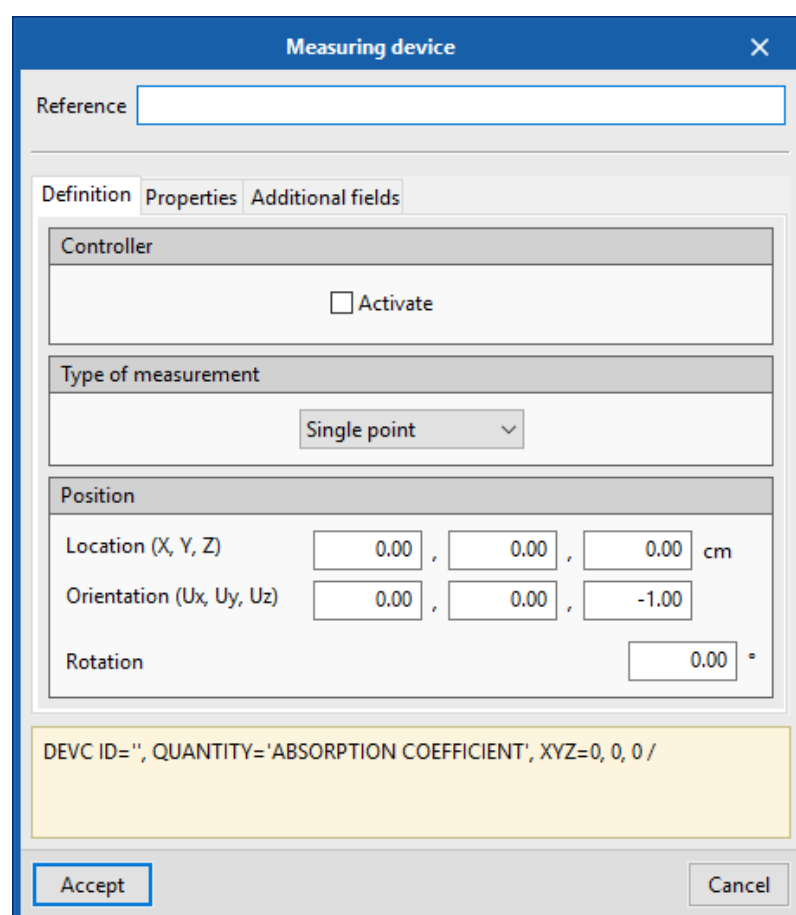
The devices can be used to record the value of a specific parameter at a point in the simulation or to represent the mathematical model of a complex sensor, such as smoke detectors, sprinklers or thermocouples.

## 3.13.1 *Gas phase measuring devices*

### 3.13.1.1 *Definition*

Within this tab, the general parameters that define the location and activation of the gaseous phase measurement device are specified.

- **Controller.** FDS controls the presence of a gaseous phase measurement device in the simulation depending on the response of a control or device.
- **Type of measurement.** Indicates whether the user wants to perform a "Spot" measurement or obtain an "Integrated quantity" of a plane or volume of the space.
- **Position.** Indicates the coordinates corresponding to the location of the gaseous phase measurement device in the scene, as well as the components of its direction vector and its rotation.



Measuring device

Reference

Definition Properties Additional fields

Controller

☐ Activate

Type of measurement

Single point

Position

Location (X, Y, Z) 0.00 , 0.00 , 0.00 cm

Orientation (Ux, Uy, Uz) 0.00 , 0.00 , -1.00

Rotation 0.00 °

DEVIC ID=", QUANTITY='ABSORPTION COEFFICIENT', XYZ=0, 0, 0 /

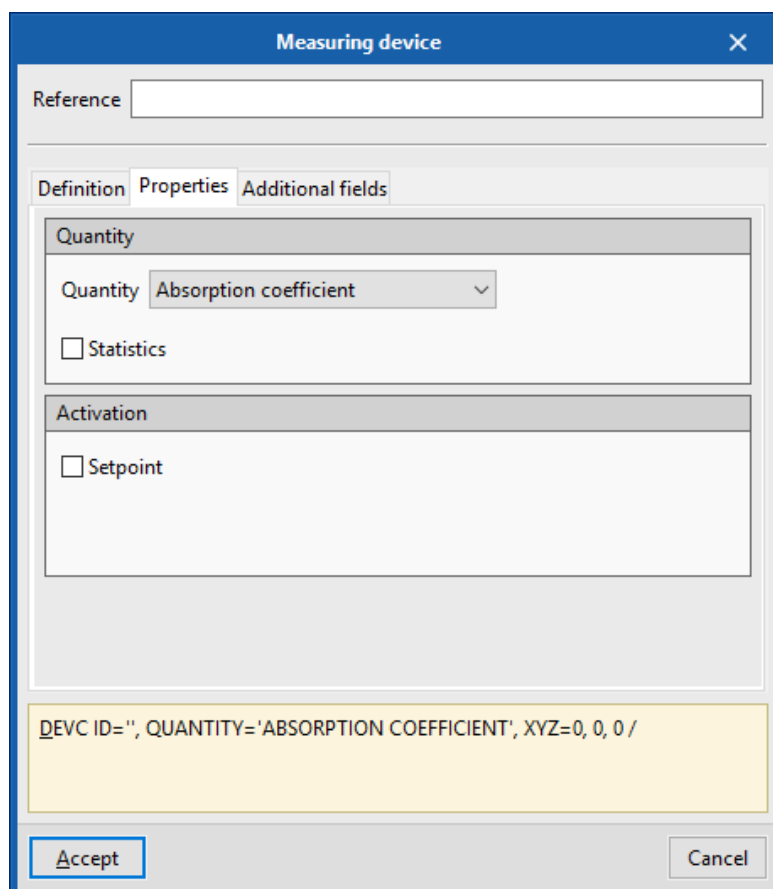
Accept Cancel

Fig. 50. Gas phase measuring device, definition

### 3.13.1.2 *Properties*

Within this group of parameters are the characteristics that define the general behaviour of the gaseous phase measurement device.

- **Quantity of measurement.** Indicates the "Quantity" that the gaseous phase measuring device must measure during the simulation. It can activate the option "Statistical results" in order to obtain a statistical value of the quantity measured (Maximum value, Minimum value, Average value ...).
- **Activation.** Indicates a value as "Control point" that indicates when the gas phase measuring device changes state.
  - **Allow a single change of state.** When activating this option, the gaseous phase measurement device can only change state once in the simulation.
  - **Initially activated.** When activating this option, it indicates that the gas phase measurement device is activated at the beginning of the simulation.



Measuring device

Reference

Definition Properties Additional fields

Quantity

Quantity Absorption coefficient

☐ Statistics

Activation

☐ Setpoint

DEVC ID=", QUANTITY='ABSORPTION COEFFICIENT', XYZ=0, 0, 0 /

Accept Cancel

Fig. 51. Gas phase measuring device, properties

## 3.13.2 *Solid phase measurement devices*

### 3.13.2.1 *Definition*

Within this tab, the general parameters that define the location and activation of the solid phase measurement device are required.

- **Controller.** FDS controls the presence of a solid phase measurement device in the simulation depending on the response of a control or device.
- **Type of measurement.** Indicates whether the user wants to perform a "Spot" measurement or obtain an "Integrated quantity" of a plane or volume of the solid.
- **Position.** Indicates the coordinates corresponding to the location of the solid phase measurement device in the scene, as well as the "Normal direction to the surface of the solid".

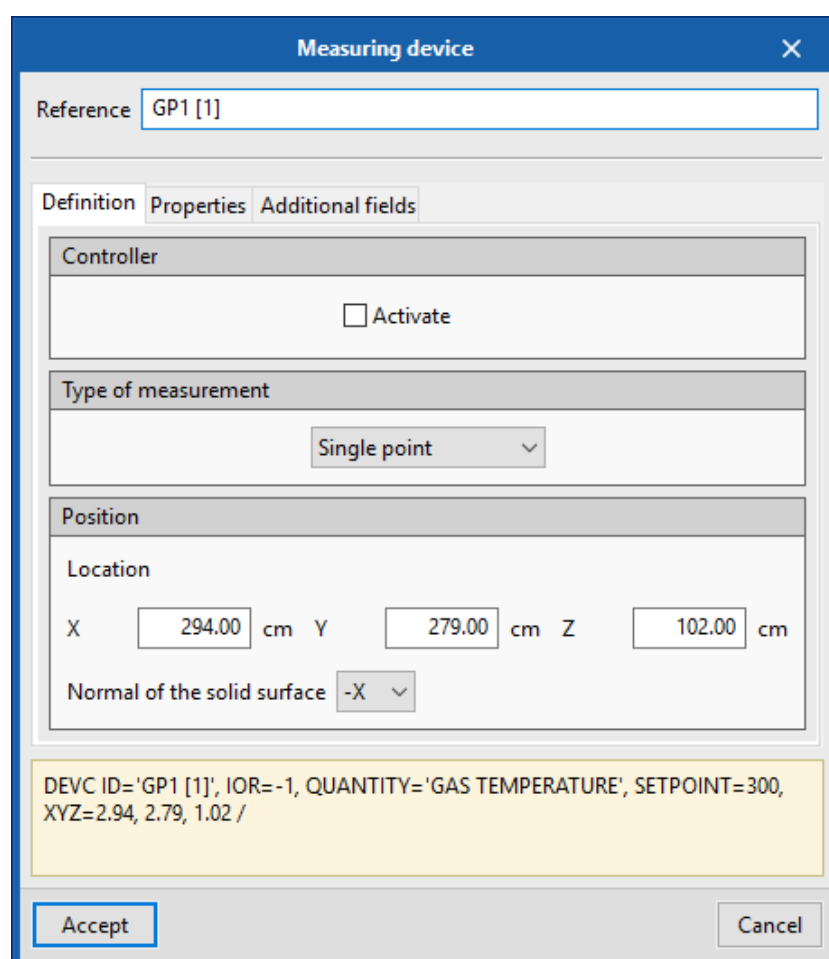
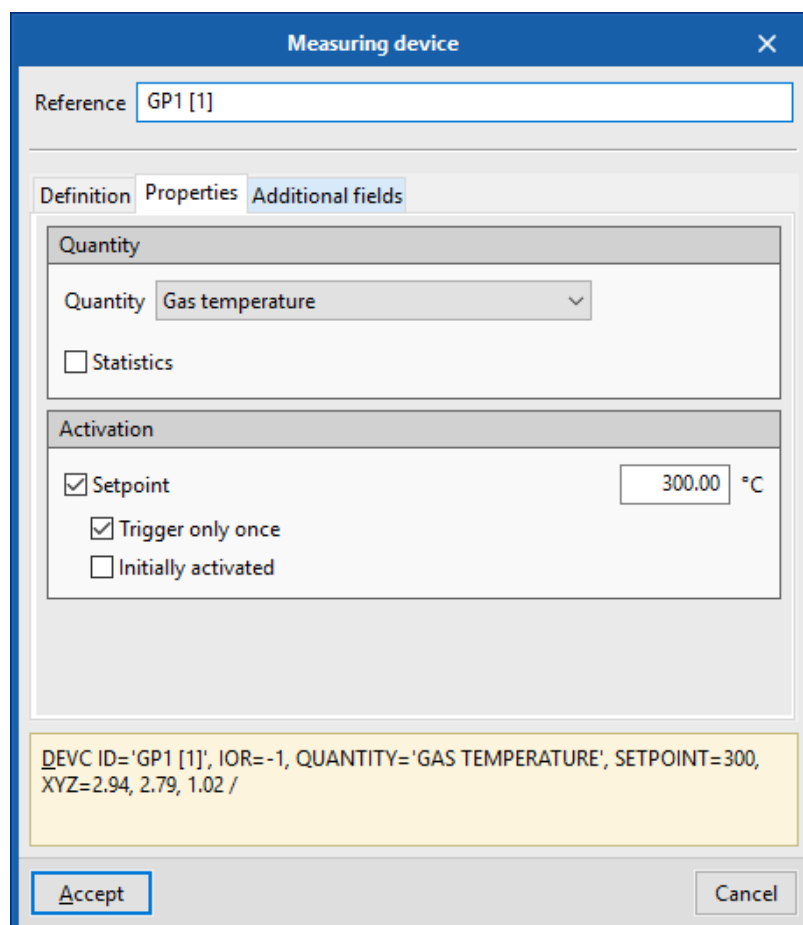


Fig. 52. Solid phase measuring device, definition

### 3.13.2.2 *Properties*

Within this group of parameters are the characteristics that define the general behaviour of the solid phase measurement device.

- **Quantity of measurement.** Indicates the "Quantity" that the solid phase measurement device must measure during the simulation. It is possible to activate the option, "Statistical results", with the purpose of obtaining a statistical value of the measured quantity (Surface integral).
- **Activation.** Indicates a value as "Control point", which show users when the solid phase measurement device changes state.
  - **Allow a single change of state.** When activating this option, the solid phase measurement device can only change status once in the simulation.
  - **Initially activated.** When activating this option, the solid phase measurement device is activated at the beginning of the simulation.



Measuring device

Reference: GP1 [1]

Definition Properties Additional fields

Quantity

Quantity: Gas temperature

☐ Statistics

Activation

☒ Setpoint 300.00 °C

☒ Trigger only once

☐ Initially activated

DEVC ID='GP1 [1]', IOR=-1, QUANTITY='GAS TEMPERATURE', SETPOINT=300, XYZ=2.94, 2.79, 1.02 /

Accept Cancel

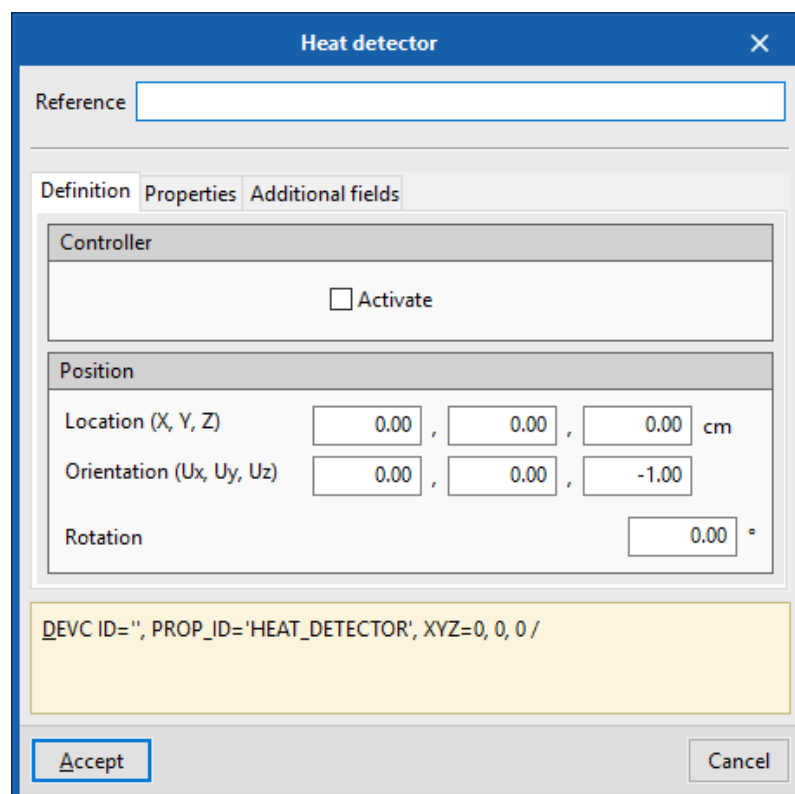
Fig. 53. Solid phase measuring device, properties

### 3.13.3 Heat detectors

#### 3.13.3.1 Definition

Within this tab, the general parameters that define the location and activation of the heat detector are required.

- **Controller.** FDS controls the presence of a heat detector in the simulation depending on the response of a control or device.
- **Position.** Indicates the coordinates corresponding to the location of the heat detector in the scene, as well as the components of its direction vector and its rotation.



The dialog box titled "Heat detector" contains the following fields and controls:

- Reference:** A text input field.
- Definition tab:**
  - Controller:** A section containing an "Activate" checkbox.
  - Position:** A section containing:
    - Location (X, Y, Z):** Three input fields with values 0.00, 0.00, and 0.00, followed by "cm".
    - Orientation (Ux, Uy, Uz):** Three input fields with values 0.00, 0.00, and -1.00.
    - Rotation:** An input field with value 0.00, followed by a degree symbol (°).
- Preview:** A yellow box showing the generated FDS command: `DEVC ID="", PROP_ID='HEAT_DETECTOR', XYZ=0, 0, 0 /`
- Buttons:** "Accept" and "Cancel" buttons at the bottom.

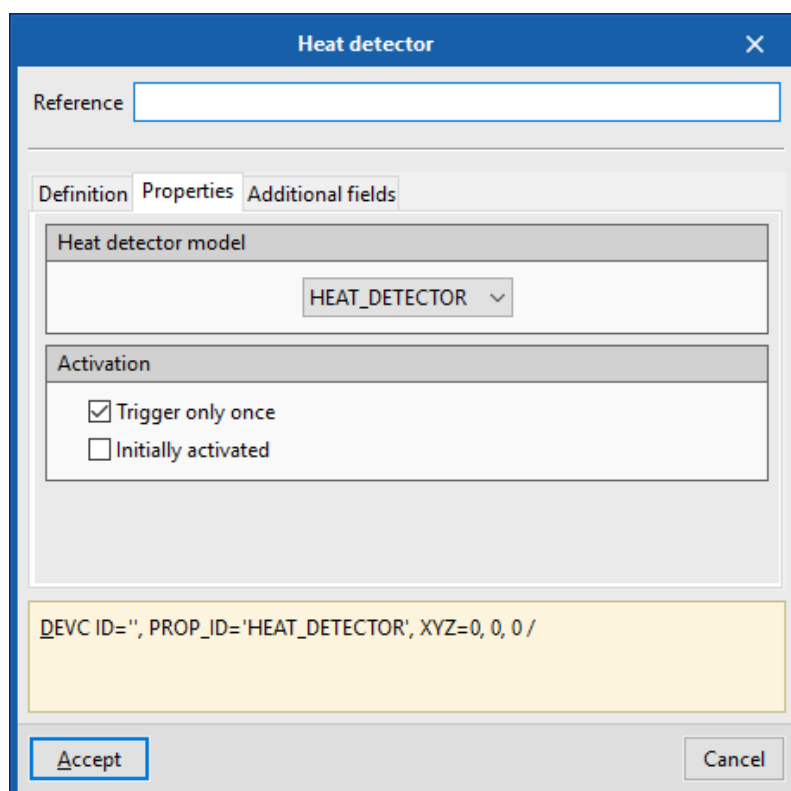
Fig. 54. Heat detector, definition

#### 3.13.3.2 Properties

Within this group of parameters are the characteristics that define the general behaviour of the heat detector.

- **Model of heat detector.** Indicates the "Heat detector model", that was previously defined, whose properties will be applied to the device.

- **Activation.**
  - **Allow a single change of state.** Activating this option allows the heat detector to only change state once in the simulation.
  - **Initially activated.** This option activates the heat detector at the beginning of the simulation.



The image shows a software dialog box titled "Heat detector". It has a close button (X) in the top right corner. Below the title bar is a "Reference" text field. Underneath are three tabs: "Definition", "Properties", and "Additional fields". The "Properties" tab is currently selected. Inside this tab, there is a section titled "Heat detector model" containing a dropdown menu with "HEAT\_DETECTOR" selected. Below that is an "Activation" section with two checkboxes: "Trigger only once" (which is checked) and "Initially activated" (which is unchecked). At the bottom of the dialog, there is a yellow-highlighted text area containing the code: `DEVIC ID=" , PROP_ID='HEAT_DETECTOR', XYZ=0, 0, 0 /`. At the very bottom are "Accept" and "Cancel" buttons.

Fig. 55. Heat detector, properties

## 3.13.4 *Beam detectors*

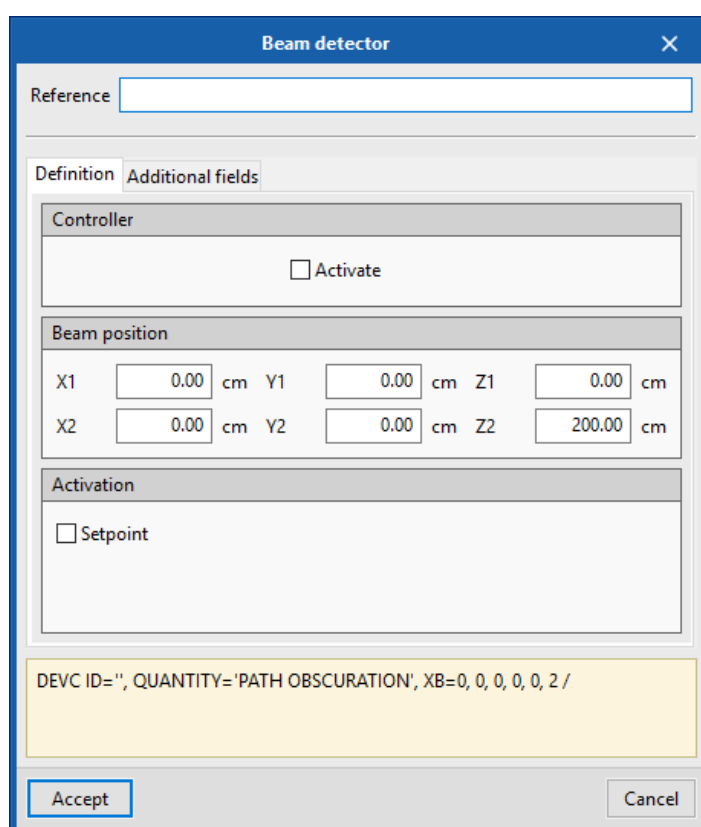
### 3.13.4.1 *Definition*

Within this tab, the general parameters that define the location and activation of the beam detector are required.

- **Controller.** FDS controls the presence of a beam detector in the simulation depending on the response of a control or device.
- **Beam position.** Indicates the coordinates corresponding to the location of the beam detector in the scene, as well as the components of its direction vector and its rotation.



- **Activation.** Showcases a value as "Control point" that indicates when the beam detector changes state.
  - **Allow a single change of state.** When activating this option, the beam detector can only change state once in the simulation.
  - **Initially activated.** Activating this option indicates that the beam detector is activated at the beginning of the simulation.



The image shows a 'Beam detector' dialog box with a blue title bar and a close button (X). It contains several sections: a 'Reference' text field, a 'Definition' tab (selected) and an 'Additional fields' tab, a 'Controller' section with an 'Activate' checkbox, a 'Beam position' section with input fields for X1, Y1, Z1, X2, Y2, and Z2 (all in cm), an 'Activation' section with a 'Setpoint' checkbox, and a yellow-highlighted text area at the bottom containing the string 'DEVC ID=", QUANTITY="PATH OBSCURATION", XB=0, 0, 0, 0, 2 /'. At the bottom are 'Accept' and 'Cancel' buttons.

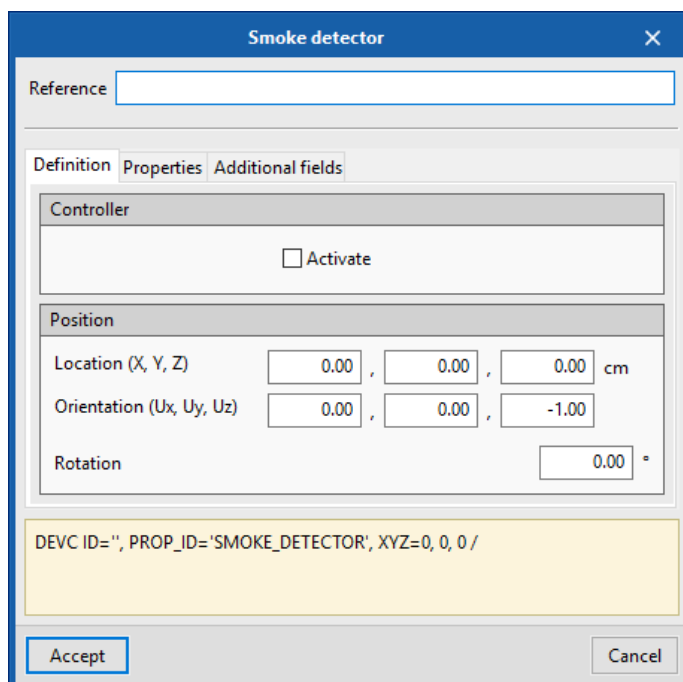
Fig. 56. Beam detector

## 3.13.5 *Smoke detectors*

### 3.13.5.1 *Definition*

Within this tab, the general parameters that define the location and activation of the smoke detector are required.

- **Controller.** FDS controls the presence of a smoke detector in the simulation depending on the response of a control or device.
- **Position.** Indicates the coordinates corresponding to the location of the smoke detector in the scene, as well as the components of its direction vector and its rotation.



The dialog box is titled "Smoke detector". It has a "Reference" field at the top. Below it are three tabs: "Definition", "Properties", and "Additional fields". The "Definition" tab is active. It contains a "Controller" section with an "Activate" checkbox. Below that is a "Position" section with fields for "Location (X, Y, Z)" (0.00, 0.00, 0.00 cm), "Orientation (Ux, Uy, Uz)" (0.00, 0.00, -1.00), and "Rotation" (0.00 °). At the bottom, a yellow box displays the text: "DEV ID= ", PROP\_ID='SMOKE\_DETECTOR', XYZ=0, 0, 0 /". There are "Accept" and "Cancel" buttons at the bottom.

Fig. 57. Smoke detector, definition

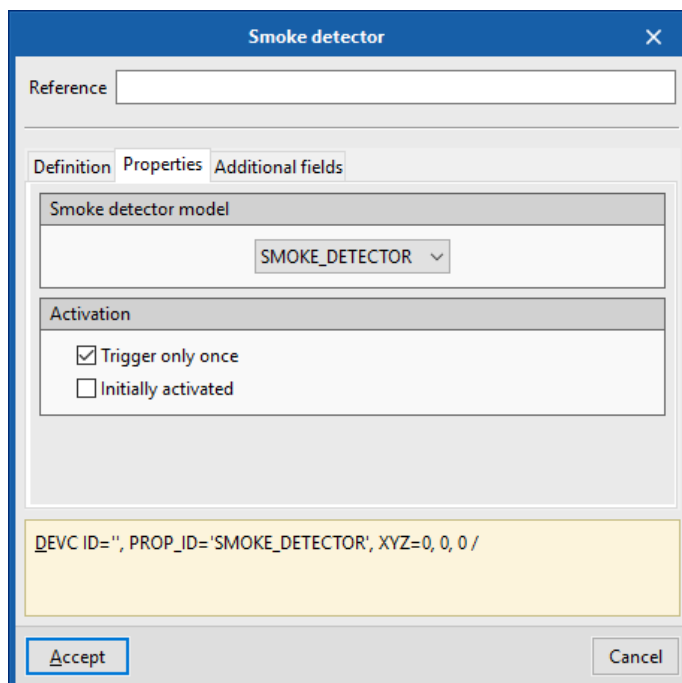
### 3.13.5.2 *Properties*

Within this group of parameters are the characteristics that define the general behaviour of the smoke detector.

- **Model of smoke detector.** Indicates that the "Smoke detector model", as previously defined, has properties that will be applied to the device.

- **Activation.**

- **Allow a single change of state.** When activating this option, the smoke detector can only change status once in the simulation.
- **Initially activated.** When activating this option, the smoke detector is activated at the beginning of the simulation.



The dialog box is titled "Smoke detector". It has a "Reference" field at the top. Below it are three tabs: "Definition", "Properties", and "Additional fields". The "Properties" tab is active. It contains a "Smoke detector model" section with a dropdown menu showing "SMOKE\_DETECTOR". Below that is an "Activation" section with two checkboxes: "Trigger only once" (checked) and "Initially activated" (unchecked). At the bottom, a yellow box displays the text: "DEV ID= ", PROP\_ID='SMOKE\_DETECTOR', XYZ=0, 0, 0 /". There are "Accept" and "Cancel" buttons at the bottom.

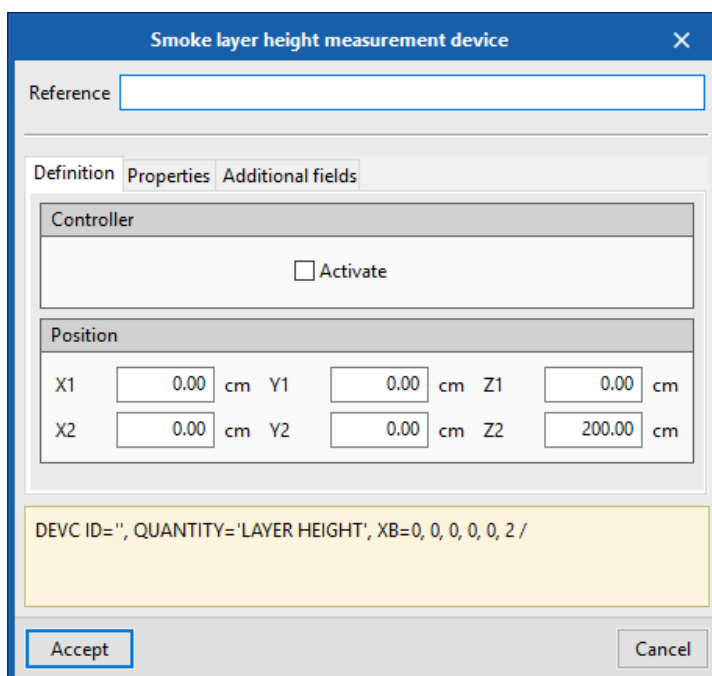
Fig. 58. Smoke detector, properties

## 3.13.6 *Smoke layer height meters*

### 3.13.6.1 *Definition*

Within this tab, the general parameters that define the location and activation of the height meter of the smoke layer are specified.

- **Controller.** FDS controls the presence of a height meter of the smoke layer in the simulation depending on the response of a control or device.
- **Position.** Indicates the coordinates corresponding to the location of the height meter of the smoke layer in the scene.



The dialog box is titled "Smoke layer height measurement device" and has a close button (X) in the top right corner. It contains a "Reference" text field at the top. Below it are three tabs: "Definition" (selected), "Properties", and "Additional fields". The "Definition" tab is divided into two sections: "Controller" and "Position". The "Controller" section has an "Activate" checkbox. The "Position" section contains two rows of coordinate fields: X1, Y1, Z1 for the first row and X2, Y2, Z2 for the second row. Each field has a numerical input box and a unit dropdown set to "cm". The values are: X1=0.00, Y1=0.00, Z1=0.00, X2=0.00, Y2=0.00, Z2=200.00. At the bottom of the dialog is a yellow-highlighted text area containing the command: `DEVC ID="", QUANTITY='LAYER HEIGHT', XB=0, 0, 0, 0, 0, 2 /`. At the very bottom are "Accept" and "Cancel" buttons.

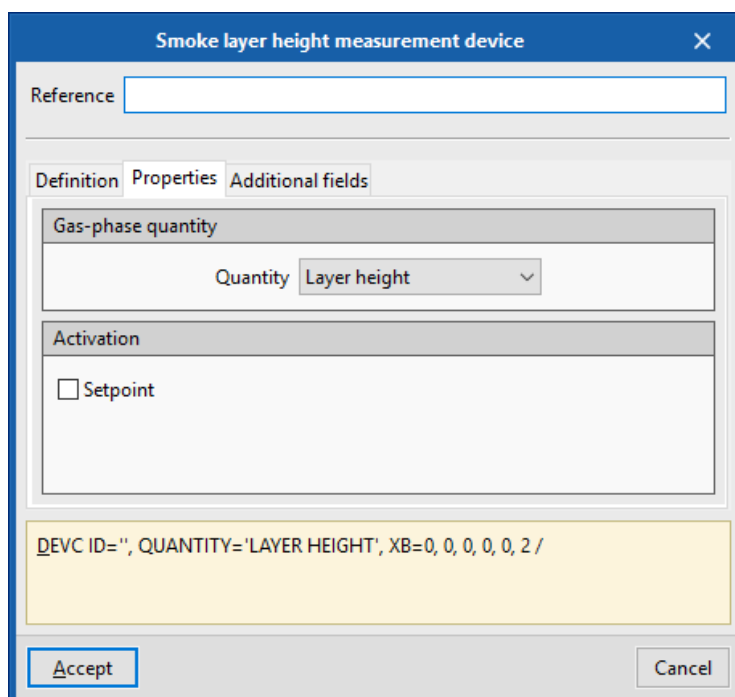
Fig. 59. Smoke layer height measurement device, definition

### 3.13.6.2 *Properties*

Within this group of parameters are the characteristics that define the general behaviour of the height meter of the smoke layer.

- **Quantity of measurement.** Indicates the quantity to be measured by the height meter of the smoke layer (layer height, maximum temperature or minimum temperature).

- **Activation.** Showcases a value as "Control point" that indicates when the height meter of the smoke layer changes state.
  - **Allow a single change of state.** When activating this option, the height meter of the smoke layer can only change status once in the simulation.
  - **Initially activated.** When activating this option, the height meter of the smoke layer is activated at the beginning of the simulation.



The dialog box is titled "Smoke layer height measurement device". It has a "Reference" field at the top. Below it are three tabs: "Definition", "Properties", and "Additional fields". The "Properties" tab is selected. It contains a "Gas-phase quantity" section with a "Quantity" dropdown set to "Layer height". Below that is an "Activation" section with a checkbox labeled "Setpoint". At the bottom, there is a yellow box containing the text: "DEVC ID=", QUANTITY='LAYER HEIGHT', XB=0, 0, 0, 0, 0, 2 /". There are "Accept" and "Cancel" buttons at the bottom right.

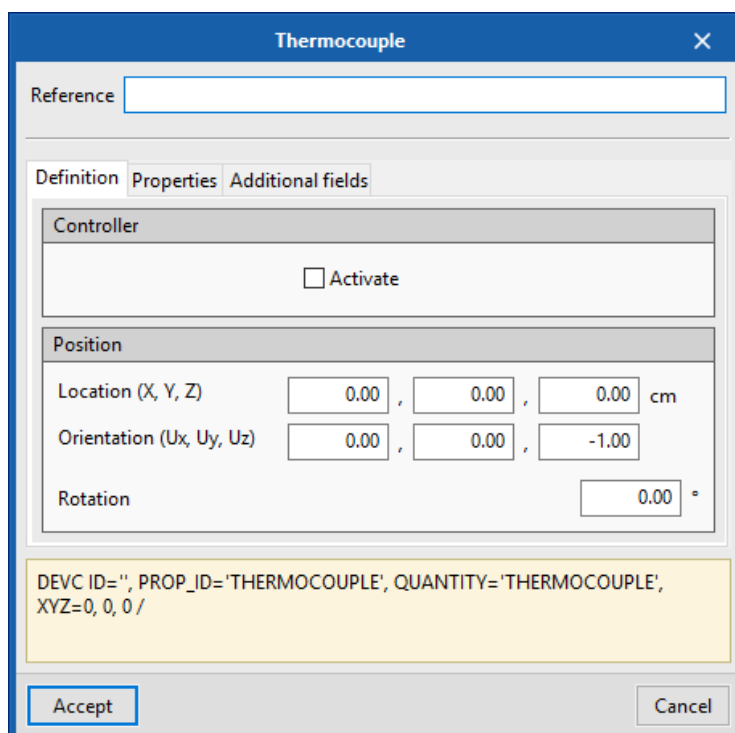
Fig. 60. Smoke layer height measurement device, properties

## 3.13.7 Thermocouples

### 3.13.7.1 Definition

Within this tab, the general parameters that define the location and activation of the thermocouple are required.

- **Controller.** FDS controls the presence of a thermocouple in the simulation depending on the response of a control or device.
- **Position.** Indicates the coordinates corresponding to the location of the thermocouple in the scene, as well as the components of its direction vector and its rotation.



The dialog box is titled "Thermocouple". It has a "Reference" field at the top. Below it are three tabs: "Definition", "Properties", and "Additional fields". The "Definition" tab is selected. It contains a "Controller" section with a checkbox labeled "Activate". Below that is a "Position" section with fields for "Location (X, Y, Z)" (0.00, 0.00, 0.00 cm), "Orientation (Ux, Uy, Uz)" (0.00, 0.00, -1.00), and "Rotation" (0.00 °). At the bottom, there is a yellow box containing the text: "DEVC ID=", PROP\_ID='THERMOCOUPLE', QUANTITY='THERMOCOUPLE', XYZ=0, 0, 0 /". There are "Accept" and "Cancel" buttons at the bottom right.

Fig. 61. Thermocouple, definition

### 3.13.7.2 *Properties*

Within this group of parameters are the characteristics that define the general behaviour of the thermocouple.

- **Thermocouple model.** Indicates that the "Thermocouple model", which was previously defined, has properties that will be applied to the device.
- **Activation.** Showcases a value as a "Control point" which indicates when the thermocouple changes state.
  - **Allow a single change of state.** When activating this option, the thermocouple can only change its status once in the simulation.
  - **Initially activated.** When activating this option, the thermocouple is activated at the beginning of the simulation.

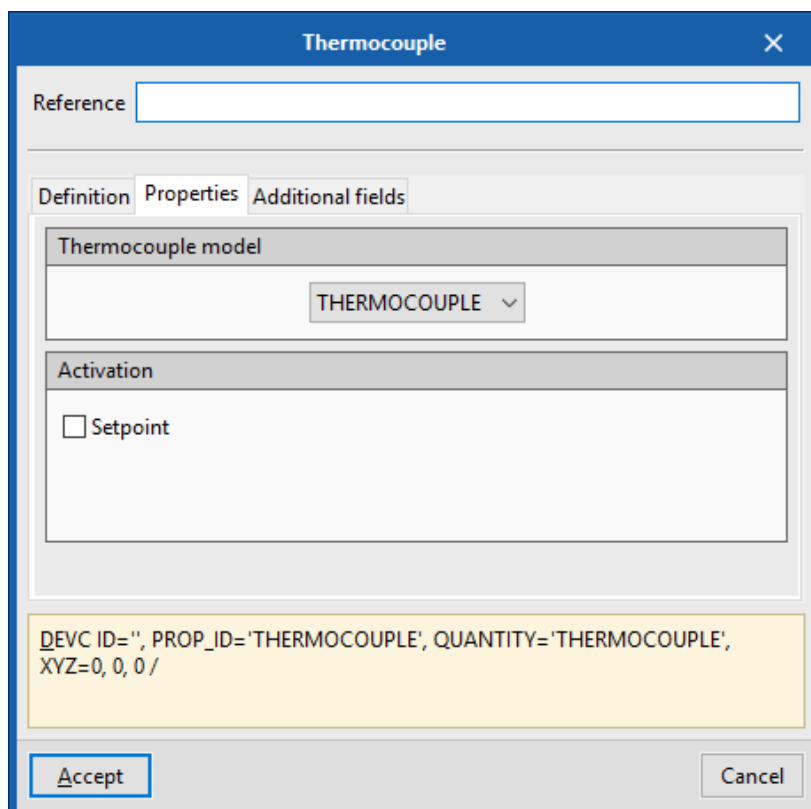


Fig. 62. Thermocouple, properties

## 3.13.8 Sprinklers

### 3.13.8.1 Definition

- **Controller.** FDS controls the presence of a sprinkler in the simulation depending on the response of a control or device.
- **Position.** It indicates the coordinates corresponding to the location of the sprinkler in the scene, as well as the components of its direction vector and its rotation.

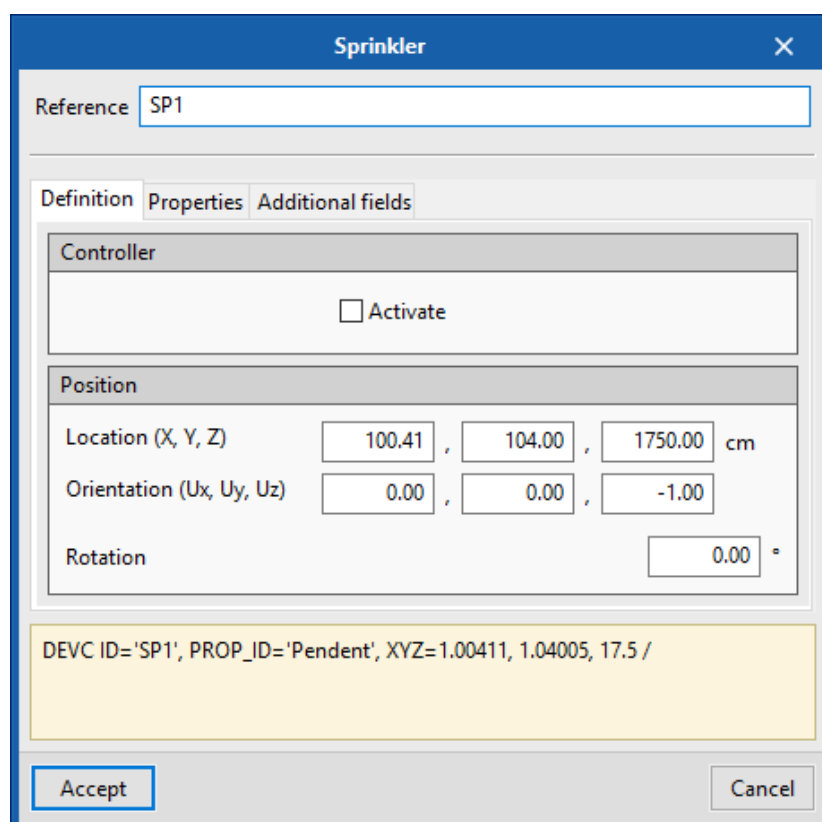
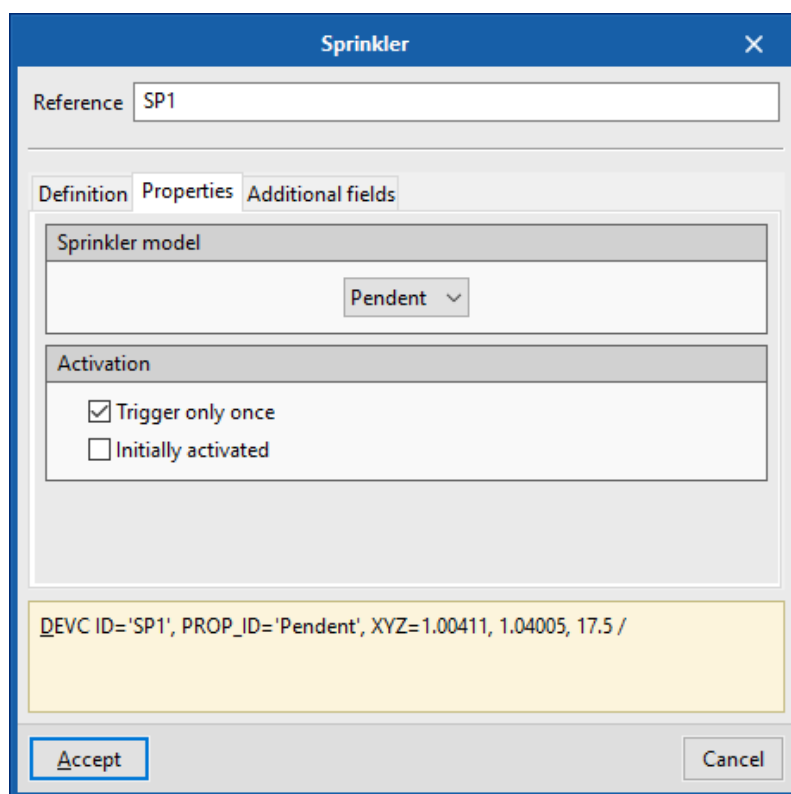


Fig. 63. Sprinkler, definition

### 3.13.8.2 Properties

- **Sprinkler model.** Indicates that the "Sprinkler Model", which was previously defined, has properties that will be applied to the device.
- **Activation.** It is possible to limit the logical state of the device to a single change, so that the sprinkler can be put into operation but not stopped or vice versa. In addition, it can be indicated that the device is activated at the beginning of the simulation.



**Sprinkler**

Reference: SP1

Definition Properties Additional fields

Sprinkler model

Pendent

Activation

☒ Trigger only once

☐ Initially activated

DEVIC ID='SP1', PROP\_ID='Pendent', XYZ=1.00411, 1.04005, 17.5 /

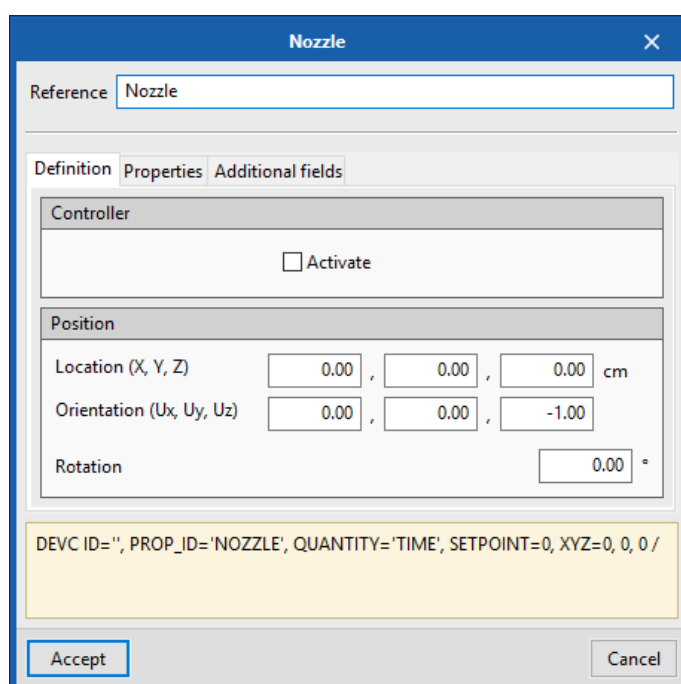
Accept Cancel

Fig. 64. Sprinkler, properties

## 3.13.9 *Nozzles*

### 3.13.9.1 *Definition*

- **Controller.** FDS controls the presence of a nozzle in the simulation depending on the response of a control or device.
- **Position.** It indicates the coordinates corresponding to the location of the nozzle in the scene, as well as the components of its direction vector and its rotation.



**Nozzle**

Reference: Nozzle

Definition Properties Additional fields

Controller

☐ Activate

Position

Location (X, Y, Z) 0.00 , 0.00 , 0.00 cm

Orientation (Ux, Uy, Uz) 0.00 , 0.00 , -1.00

Rotation 0.00 °

DEVIC ID="", PROP\_ID='NOZZLE', QUANTITY='TIME', SETPOINT=0, XYZ=0, 0, 0 /

Accept Cancel

Fig. 65. Nozzle, definition

### 3.13.9.2 *Properties*

- **Nozzle model.** Indicates that the "Mouthpiece Model", which was previously defined, has properties that will be applied to the device.
- **Activation.** Indicates when the nozzle in the simulation will be activated.

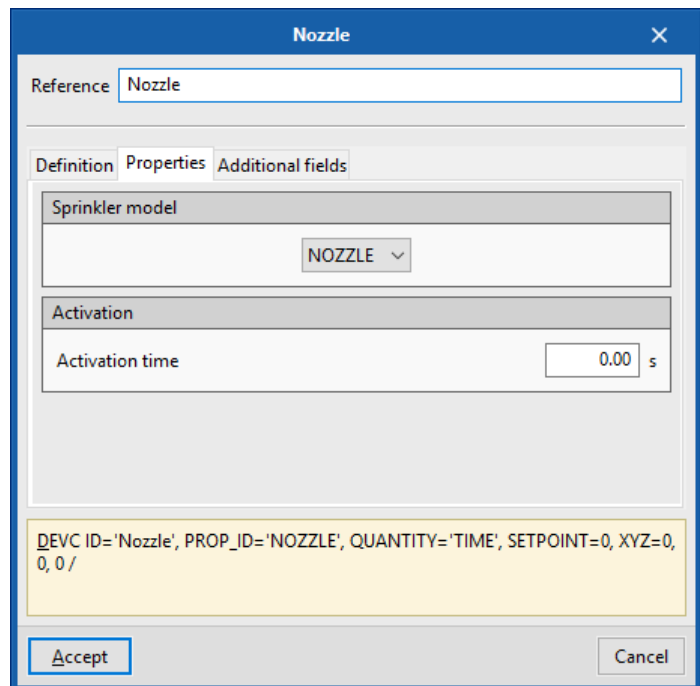


Fig. 66. Nozzle, properties

## 3.14 Controls

The controls are mechanisms that allow the users to describe behaviours with greater complexity than with the devices.

- **Function.** Indicates the type of control function.
  - **ANY.** The status of the control changes if any of the devices or controls defined as "Input variables" is activated.
  - **ALL.** The status of the control changes if all the devices or controls defined as "Input variables" is activated.

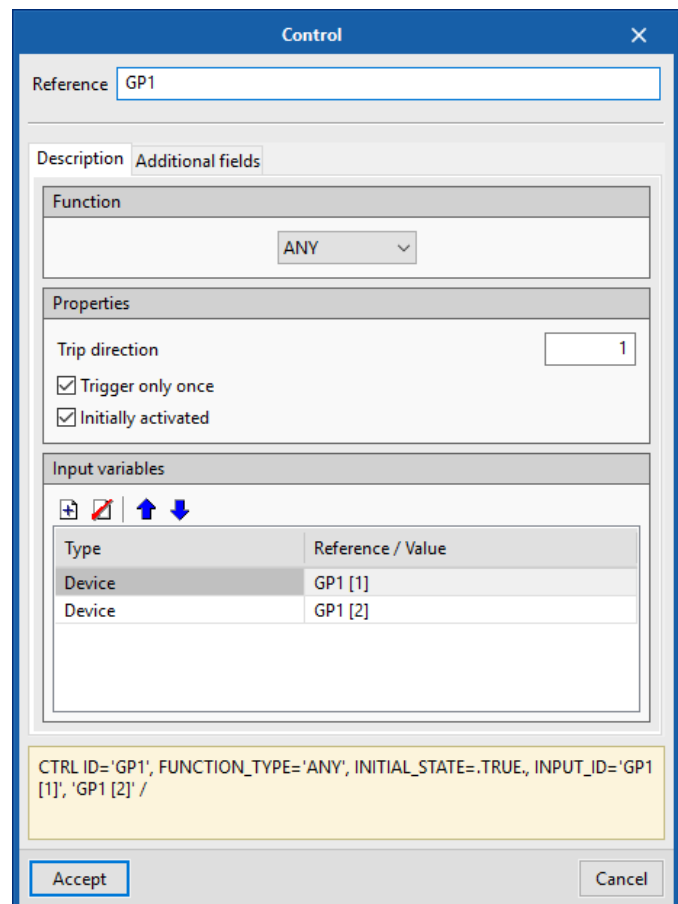


Fig. 67. Control



- **ONLY.** The status of the control changes if only N of the devices or controls defined as "Input variables" are activated. The value of N is entered through the field "Minimum number of variables".
- **AT\_LEAST.** The control status changes if at least N of the devices or controls defined as "Input variables" are activated. The value of N is entered through the field "Minimum number of variables".
- **TIME\_DELAY.** The state of the control changes after a time interval ("Delay time") since the device or control defined as "Input variable" is activated.
- **CUSTOM.** The status of the control changes based on the result of the ramp function entered as "Input variable".
- **DEADBAND.** The state of the control changes based on the value of the device defined as "Input variable" in a manner analogous to the operation of a thermostat. If the value of the "Dimension" parameter is "Lower", the status of the control will change when the value of the "Input variable" falls below the "Lower control point". The opposite operation will occur when the value of "Dimension" is "Superior".
- **KILL.** The simulation will be interrupted when the device or control defined as "Input variable" is activated.
- **RESTART.** A restart file will be generated when the device or control defined as "Input variable" is activated. This way, the simulation can be restarted later from this point.
- **SUM.** The value of the control is the sum of the devices, controls and, optionally, the constant defined as "Input variables".
- **SUBSTRACT.** The value of the control is the subtraction of the device, control or constant defined as "Subtracting" the value of the device, control or constant defined as "Minuend".
- **MULTIPLY.** The value of the control is the multiplication of the devices, controls and, optionally, the constant defined as "Input variables".
- **DIVIDE.** The value of the control is the division of the device, control or constant defined as "Divisor" to the value of the device, control or constant defined as "Dividend".
- **POWER.** The value of the control is the result of the power with base equal to the value of the device or control defined as "Base" and exponent equal to the value of the device or control defined as "Exponent".

- **EXP.** The value of the control is the exponential of the device or control defined as "Input variable".
- **LOG.** The value of the control is the natural logarithm of the device or control defined as "Input variable".
- **COS.** The value of the control is the cosine of the device or control defined as "Input variable".
- **SIN.** The value of the control is the sine of the device or control defined as "Input variable".
- **ACOS.** The value of the control is the arccosine of the device or control defined as "Input variable".
- **ASIN.** The value of the control is the arcsine of the device or control defined as "Input variable".
- **PID.** A PID control function (Proportional Integral Derivative) is a feedback controller commonly used to control electrical and mechanical systems. The function calculates an error between a process variable ("Input variable") and a desired set point ("Control point"). The purpose of the PID function is to minimize the error. A PID control function is calculated as:

$$u(t) = K_p e(t) + K_i \int_0^t e(t) dt + \frac{K_d de(t)}{dt}$$

Where:

$K_p$ : Proportional gain

$K_i$ : Integral gain

$K_d$ : Derivative gain

$e(t)$ : Target value

$u(t)$ : Output value

- **Properties.**

- **Direction of the displacement.** A positive address value indicates that the control will change its initial state when its result is higher than the control point and will have its initial state when it is lower. A negative address value will produce the opposite behaviour. The control will change its initial state when its result are lower than the control point and will have its initial state when it is higher.
- **Allow a single change of state.** When activating this option, the control can only change state once in the simulation.

- **Initially activated.** When activating this option, the control is activated at the beginning of the simulation.
- **Control point.** It is the value of the control function where the control changes state. Only functions that return a numerical value should be indicated.

## 3.15 Sections

The sections are entities of the FDS calculation engine that obtains the value of several gaseous quantities in more than one point.

### 3.15.1.1 *Description*

- **Geometry.** Indicates the measurement area as a plane perpendicular to one of the global axes of the scene and can also be specified manually. The measurement area can be a volume, a plane or a line.
- **Position.** Indicates the height of the plane, if it is a plane perpendicular to the global axes of the work, or if it is the limits of the measurement area.

### 3.15.1.2 *Properties*

- **Quantity of measurement.** Indicates the quantity to be measured by the section.
- **General properties.**
  - **Generate vectors.** By activating this option, animated vectors will be generated that can be visualized in Smokeview once the simulation is finished. If two sections are in the same plane, this option should be activated in one of them. Otherwise, an extra speed section will be generated.
  - **Obtain the data for the centre of the cells.** By default, the FDS calculation engine calculates the results of the sections in the corners of the cells. When activating this option, these values will be obtained for the centre of the cells.

**Slice** [X]

Reference: Temp. Ground floor

Description Properties Additional fields

**Geometry**

Plane perpendicular to the Z-axis ▾

**Position**

Plane height: 160.00 cm

SLCF ID= 'Temp. Ground floor', PBZ=1.6, QUANTITY='TEMPERATURE' /

Accept Cancel

**Slice** [X]

Reference: Temp. Ground floor

Description Properties Additional fields

**Gas-phase quantity**

Quantity: Temperature ▾

**General properties**

☐ Generate vectors

☐ Obtain data for cell-centres

SLCF ID= 'Temp. Ground floor', PBZ=1.6, QUANTITY='TEMPERATURE' /

Accept Cancel

Fig. 68. Slice, description and properties

## 4 Simulation

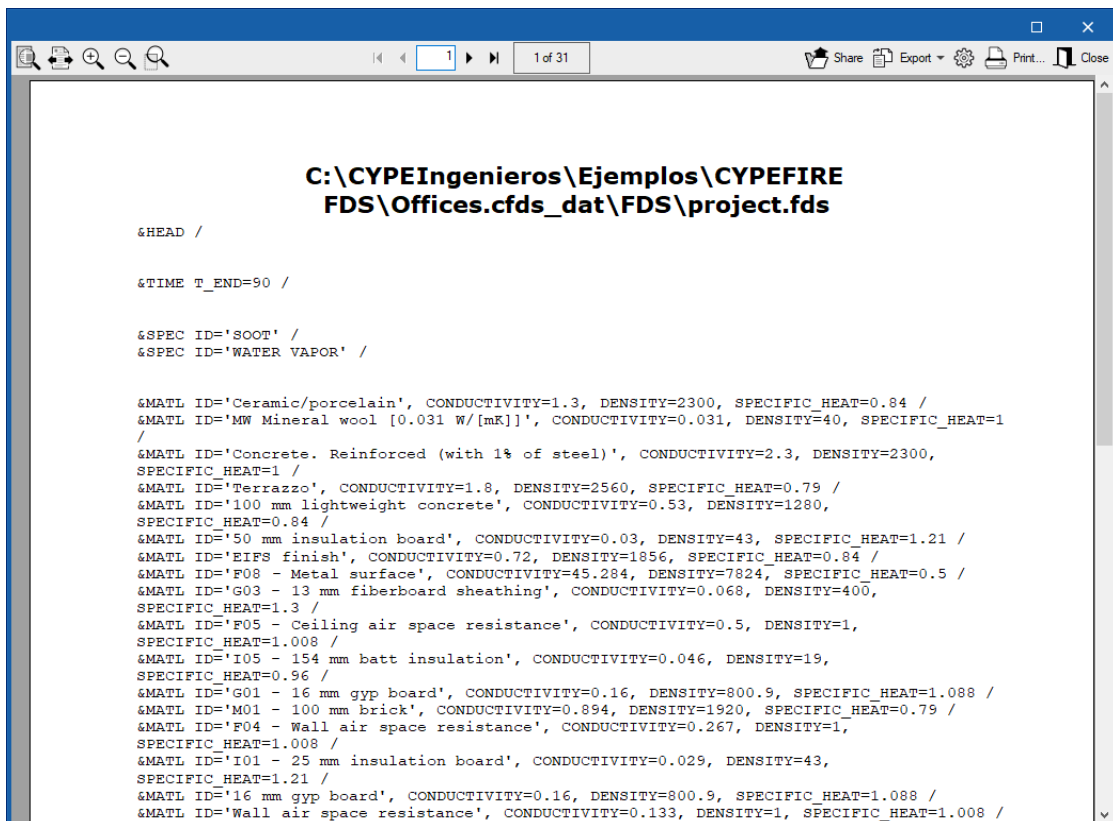
Once the introduction and definition of all the elements that make up the simulation of the fire are complete, users can turn to this section.

### 4.1 Analysis/Calculation

CYPEFIRE FDS has different tools that help users solve this section using the Analysis/Calculation that was created for the model.

#### 4.1.1 *FDS Report*

Here, the reports that have been generated and contain defined characteristics of the simulation can be obtained as a *.fds* file which can be read with any text editor. Additionally, the route that the file is located can be viewed, as shown in the image below.



The screenshot shows a text editor window with the following content:

```

C:\CYPEIngenieros\Ejemplos\CYPEFIRE
FDS\Offices.cfds_dat\FDS\project.fds

&HEAD /

&TIME T_END=90 /

&SPEC ID='SOOT' /
&SPEC ID='WATER VAPOR' /

&MATL ID='Ceramic/porcelain', CONDUCTIVITY=1.3, DENSITY=2300, SPECIFIC_HEAT=0.84 /
&MATL ID='MW Mineral wool [0.031 W/[mK]]', CONDUCTIVITY=0.031, DENSITY=40, SPECIFIC_HEAT=1 /
&MATL ID='Concrete. Reinforced (with 1% of steel)', CONDUCTIVITY=2.3, DENSITY=2300, SPECIFIC_HEAT=1 /
&MATL ID='Terrazzo', CONDUCTIVITY=1.8, DENSITY=2560, SPECIFIC_HEAT=0.79 /
&MATL ID='100 mm lightweight concrete', CONDUCTIVITY=0.53, DENSITY=1280, SPECIFIC_HEAT=0.84 /
&MATL ID='50 mm insulation board', CONDUCTIVITY=0.03, DENSITY=43, SPECIFIC_HEAT=1.21 /
&MATL ID='EIFS finish', CONDUCTIVITY=0.72, DENSITY=1856, SPECIFIC_HEAT=0.84 /
&MATL ID='F08 - Metal surface', CONDUCTIVITY=45.284, DENSITY=7824, SPECIFIC_HEAT=0.5 /
&MATL ID='G03 - 13 mm fiberboard sheathing', CONDUCTIVITY=0.068, DENSITY=400, SPECIFIC_HEAT=1.3 /
&MATL ID='F05 - Ceiling air space resistance', CONDUCTIVITY=0.5, DENSITY=1, SPECIFIC_HEAT=1.008 /
&MATL ID='I05 - 154 mm batt insulation', CONDUCTIVITY=0.046, DENSITY=19, SPECIFIC_HEAT=0.96 /
&MATL ID='G01 - 16 mm gyp board', CONDUCTIVITY=0.16, DENSITY=800.9, SPECIFIC_HEAT=1.088 /
&MATL ID='M01 - 100 mm brick', CONDUCTIVITY=0.894, DENSITY=1920, SPECIFIC_HEAT=0.79 /
&MATL ID='F04 - Wall air space resistance', CONDUCTIVITY=0.267, DENSITY=1, SPECIFIC_HEAT=1.008 /
&MATL ID='I01 - 25 mm insulation board', CONDUCTIVITY=0.029, DENSITY=43, SPECIFIC_HEAT=1.21 /
&MATL ID='16 mm gyp board', CONDUCTIVITY=0.16, DENSITY=800.9, SPECIFIC_HEAT=1.088 /
&MATL ID='Wall air space resistance', CONDUCTIVITY=0.133, DENSITY=1, SPECIFIC_HEAT=1.008 /

```

Fig. 69. FDS report

## 4.1.2 Check errors

This tool is useful to activate before beginning the simulation since it can take awhile for the complete simulation to finish running. Therefore, if major errors are corrected before the simulation, it can help the program perform a smoother analysis.

## 4.1.3 Smokeview

Smokeview is a tool designed to visually represent the data obtained after analyzing the *.fds* file through the generated *project.smv* file. When starting the Simulation in the main toolbar, Smokeview will be launched automatically.

Smokeview is used before, during, and after the simulation of the model. Smokeview can be used before starting the simulation to verify that the model is correct ( $t = 0$  s) and during the calculation process to monitor the progress of the simulation.

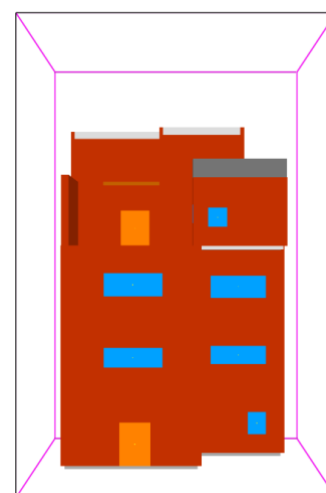


Fig. 70. Smokeview

## 4.2 Graphics

Once the simulation is finished, the graphs from the main toolbar of the program can be accessed.

In these graphs, the evolution of different parameters that are measured in the simulation (heat released, heat by radiation, heat by convection) can be observed, as seen in the following image, as well as the operation of the devices and the controls introduced.

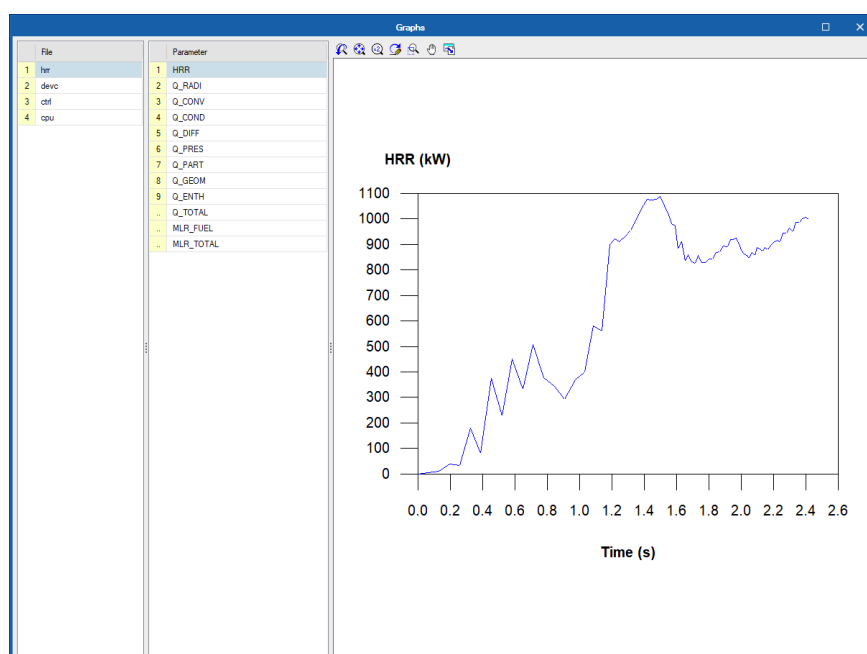


Fig. 71. Graphics display, evolution of measured parameters

## **Contact**

How to setup a project, navigate through the user interface, design within the software, and obtain results based off the design should be known after the completion of this manual for CYPEFIRE FDS. If there are still questions, issues, or further information is needed, please visit our website or contact CYPE. The contact information is below.

### **CYPE Ingenieros**

Avda. de Loring, 4  
03003 Alicante - Spain  
Tel. (+34) 965 92 25 50  
Fax (+34) 965 12 49 50  
[cype@cype.com](mailto:cype@cype.com)

### **Technical Support:**

[support@cype.com](mailto:support@cype.com)

### **North America & United Kingdom Contact:**

USA (+1) 202 569 8902  
UK (+44) 20 3608 1448  
[marketing@cype.com](mailto:marketing@cype.com)  
[www.en.cype.com](http://www.en.cype.com)

### **CYPE Italia**

Tel. (+39) 06 94 803 504  
Tel. (+39) 06 94 800 227  
[supporto.italia@cype.com](mailto:supporto.italia@cype.com)

### **CYPE France**

**Tel. (+33) 2 30 96 1744**  
**Fax (+33) 2 22 44 2508**  
[cype.france@cype.com](mailto:cype.france@cype.com)

### **CYPE em Portugal (TOP Informática, Lda.)**

Tel. (+351) 253 209 430  
[geral@top-informatica.pt](mailto:geral@top-informatica.pt)

Thank you for choosing CYPE and CYPEFIRE FDS.

## **References**

McGrattan, K., Hostikka, S., McDermott, R., Floyd, J., & Vanella, M. (2018). *Fire Dynamics Simulator Technical Reference Guide* (NIST Special Publication 1018-1 Sixth Edition ed., Vol. Mathematical Model).

McGrattan, K., Hostikka, S., McDermott, R., Floyd, J., Weinschenk, C., & Overholt, K. (2016). *Fire Dynamics Simulator User's Guide* (NIST Special Publication 1019 Sixth Edition ed.).