



INDIRES User Documentation Simulation of Effects of Incidents on Environment & Infrastructure

30th November 2020

**This page is
intentionally
left blank.**

TABLE of CONTENTS

1	INTENDED AUDIENCE.....	5
2	OVERVIEW	7
3	SOFTWARE DESCRIPTION.....	9
3.1	Brief Description	9
3.2	System Requirements	9
3.3	Where to Download the Software	9
3.4	Pyrosim Basics	10
3.5	Working with Files	11
3.5.1	Creating a new Pyrosim model.....	11
3.5.2	Saving a Pyrosim Model	11
3.5.3	Open a saved Pyrosim Model.....	11
3.5.4	Importing FDS models	11
3.5.5	Exporting FDS models	11
3.5.6	Importing CAD files	12
3.5.7	DWG/DXF import	12
3.5.8	Working with REVIT Architecture.....	13
3.6	Meshes	13
3.6.1	Uniform Meshes.....	13
3.6.2	Non-uniform Meshes	14
3.6.3	Multiple Meshes	15
3.7	Materials.....	15
3.7.1	Solid Materials	15
3.7.2	Liquid Fuels	16
3.8	Surfaces	16
3.8.1	Surface Creation.....	17
3.8.2	Surface Types.....	17
3.9	Geometry.....	19
3.9.1	Obstructions.....	19
3.9.2	Vents.....	20
3.10	Reactions	20
4	PRACTICAL CASE TO BUILD A MINE MODEL FOR CFD FIRE SIMULATIONS	23
4.1	Introduction	23
4.1.1	Choice of the Mine Risk Scenario	23
4.1.2	Modelling the Mine Geometry	23
4.1.3	Modelling the Fire.....	23
4.1.4	Assessing the Results	23

4.2	Input Parameters to Define the Mine Risk Scenario	23
4.2.1	Accidents with Highest Likelihood.....	24
4.2.2	Materials with Highest Likelihood of Ignition.....	24
4.2.3	Fire Location	25
4.2.4	Ventilation Conditions.....	25
4.2.5	Emergency Procedure	26
4.3	CFD Model	26
4.3.1	Creation of the Mine's Geometry.....	27
4.3.2	Fire Definition	30
4.3.3	Boundary Conditions.....	33
4.3.4	Emergency Procedure	37
4.4	Results	38
4.4.1	Results Measurement Points.....	38
4.4.2	Results in a Plane of Measure	41
5	FURTHER INFORMATION	43

1 INTENDED AUDIENCE

An output of the INDIRES project is user documentation for each of the products produced during the project. However, the status of the various products differs and, for this reason, the intended audience for the *User Documentation* will be different for each product. It is necessary, therefore, to indicate the type of user who will benefit from this particular *User Document*.

The software for simulating the effects of incidents on the environment and infrastructure is fully complete. As such, the information provided in this *User Document* could be utilised by end users who have a need for this functionality. Such users can consider the instructions on how to use the software as definitive.

End users who want to know more are referred to *Section 5*.

**This page is
intentionally
left blank.**

2 OVERVIEW

This document is divided into two parts.

The first part describes the software FDS and its pre-processor Pyrosim. A detailed description is provided of all the steps needed to correctly build a model of the environmental conditions in the event of a fire in a coal mine, with particular emphasis on the individual steps.

The second part is a practical guide that presents an overview of the process of building a mine model with an actual fire of a particular type, and the final step needed to make an assessment of the results.

**This page is
intentionally
left blank.**

3 SOFTWARE DESCRIPTION

This section describes the software, FDS, and its preprocessor Pyrosim, and the steps required to build a model of a mine fire.

3.1 Brief Description

PyroSim is a graphical user interface for the Fire Dynamics Simulator (FDS). FDS models can predict smoke, temperature, carbon monoxide, and other substances during fires. The results of these simulations have been used to ensure the safety of buildings before construction, evaluate safety options of existing buildings, reconstruct fires for post-accident investigation, and assist in firefighter training.

FDS is a powerful fire simulator which was developed at the National Institute of Standards and Technology (NIST). FDS simulates fire scenarios using computational fluid dynamics (CFD) optimized for low-speed, thermally-driven flow. This approach is very flexible and can be applied to fires ranging from stove-tops to oil storage tanks. It can also model situations that do not include a fire, such as ventilation in buildings. FDS and the Smokeview visualization program are both closely integrated into PyroSim.

The PyroSim interface provides immediate feedback on input and ensures the correct format for the FDS input file. You can work in either metric or imperial units and you can switch between the two at any time. In addition, PyroSim offers high-level 2D and 3D geometry creation features, such as diagonal walls, background images for sketching, object grouping, flexible display options, as well as copying and replication of obstructions. You can import DXF files that include either 3D faces or 2D lines that can be extruded to create 3D objects in PyroSim.

3.2 System Requirements

PyroSim runs on the Microsoft Windows operating system. You should have at least 256MB of system RAM, and a graphics card that supports OpenGL 1.1 or later. If you want to perform realistic simulations, we recommend 2GB of system RAM, a graphics card with at least 128MB of graphics memory, and a dual or quad core processor. PyroSim supports parallel processing on multi-core/multi-processor computers.

3.3 Where to Download the Software

The FDS software is a free software that is downloadable from the website <https://pages.nist.gov/fds-smv/downloads.html>.

In order to make the use of FDS easier, there is a preprocessor called Pyrosim, which is more friendly and allows the user to easily model the mine's features. Pyrosim can be downloaded from the website <https://support.thunderheadeng.com/pyrosim/>, and, although this is not free software, there is a 30-day trial free version.

There is no difference in the performance of the licensed version and the 30-day trial version.

3.4 Pyrosim Basics

PyroSim provides four editors for the fire model: the 3D View, 2D View, Navigation View, and the Record View, as shown in *Figure 1*. The 3D and 2D view can be selected on the screen, in the bottom-left corner.

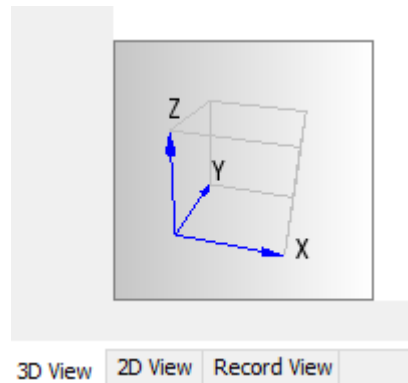


Figure 1 – Pyrosim Views

- **Navigation View:** This view lists many of the important records in the model. It allows you to organize your model geometry into groups such as a conveyor belt or an obstacle, and modifying records is often faster and easier in this view.
- **3D View:** This view shows a 3D representation of the current fire model. You can explore the model using different view controls. You can also control the appearance of the model with options like smooth shading, textures, and object outlines. Geometric features can also be changed.
- **2D View:** This view is useful for quickly sketching geometry such as walls and furniture. You can choose from three viewing planes and perform many useful geometric manipulations
- **Record View:** This view gives a preview of the FDS input file that will be generated for the simulation. It also provides a way to add custom records that will not be processed by PyroSim, but will be sent to FDS.

There are several tools that can be used to navigate the model and select objects. The tools for the 3D view are found in the navigation toolbar above the 3D view as shown in *Figure 2*.



Figure 2 – Pyrosim Toolbar

- **Select/Manipulate Tool:** This general-purpose tool can be used to perform most navigation activities.
- **Orbit Tool:** This is another general-purpose tool that may be more familiar to existing PyroSim users, but it is more limited than the Select/Manipulate Tool.

- **Roam Tool:** This tool allows the user to move into the model rather than viewing it only from the outside. This tool can take some experimentation, but once mastered, it can provide unique views of the model.
- **Pan Tool, Zoom Tool, Zoom Box Tool:** These tools break out the functionality of the above tools so that dragging any mouse button will perform the required action.

3.5 Working with Files

Several files are used when performing a fire analysis using PyroSim. These include the PyroSim model file, the FDS input file, and FDS output files. This section describes how to load and save files in the formats supported by PyroSim.

3.5.1 Creating a new Pyrosim model

When PyroSim is started, it begins with an empty model. You can close the current model and create a new empty model by opening the File menu and clicking New. PyroSim always has one (and only one) active model.

3.5.2 Saving a Pyrosim Model

To save a new model: On the File menu and click Save.

3.5.3 Open a saved Pyrosim Model

PyroSim model files have a PSM extension. To open a saved model:

- On the File menu, click Open.
- Select the file and click the Open button.

3.5.4 Importing FDS models

PyroSim allows you to import existing FDS input files. When you import an FDS file, PyroSim will create a new PyroSim model from the imported file. During import, PyroSim will check for the validity of each record. If errors are detected, you will be notified.

To import existing FDS models into PyroSim:

- On the File menu, select Import FDS/CAD File or click the Import button on the main toolbar.
- Select the FDS file and click Open.

3.5.5 Exporting FDS models

PyroSim also allows you to explicitly export the current model to an FDS input file. You can manually edit the file to take advantage of advanced FDS features, or to easily transfer the input file to a different machine or special version of FDS.

To export an FDS file:

- On the File menu, click Export, then click FDS File or click the Export button on the main toolbar.
- Enter the file name and click Save.

3.5.6 Importing CAD files

PyroSim allows users to import several types of CAD file formats, including the DXF and DWG format and the STL (Standard Tessellation Language) format. Each type of file provides a variety of geometry that can either be directly represented as obstructions or as drawing guides in the PyroSim model.

3.5.7 DWG/DXF import

DWG and DXF files are standard drawing formats supported by a variety of CAD software including Autodesk's AutoCAD and Revit software suites. These types of files can contain both 3D geometry, such as walls, slabs, solids, and faces and 2D geometry, such as circles, arcs, and lines.

To import DXF or DWG files, perform the following:

- On the File menu, select Import FDS/CAD File... or click the Import button on the main toolbar.
- Select the desired DXF or DWG file to import and click Open.
- Select the length unit from the Import Options dialogue. If the DXF/DWG file stored a length unit with the file, this will be selected by default.
- Click OK to begin import.

This is illustrated in *Figure 3*.

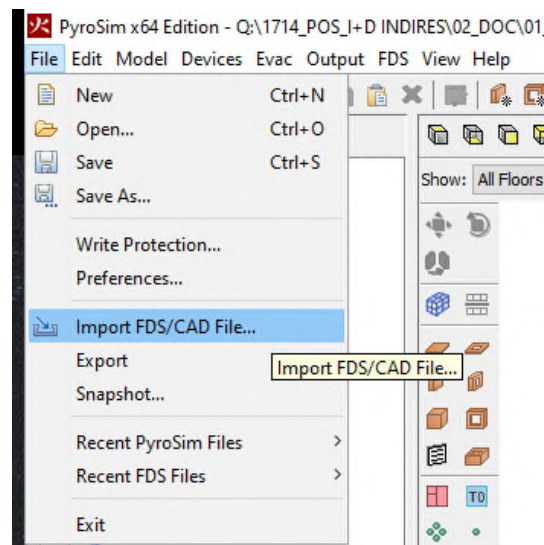


Figure 3 – Pyrosim Options to Import Files

3.5.8 Working with REVIT Architecture

While PyroSim cannot directly import Autodesk Revit files (RVT), Revit can export to DWG, which can then be imported into PyroSim.

3.6 Meshes

All FDS calculations are performed within computational meshes. Every object in the simulation (e.g. obstructions and vents) must conform to the mesh. When an object's location doesn't exactly conform to a mesh, the object is automatically repositioned during the simulation. Any object that extends beyond the boundary of the physical domain is cut off at the boundary.

FDS uses a Poisson solver based on Fast Fourier Transforms (FFTs). A side effect of this approach is that optimal mesh divisions are constrained to the form $2^u 3^v 5^w$, where u , v and w are integers. For example, $64 = 2^6$, $72 = 2^3 \times 3^2$, and $108 = 2^2 \times 3^3$ are good mesh dimensions. However, 37, 99 and 109 are not. In addition, using a prime number of cells along an axis may cause undesirable results. PyroSim warns when the number of divisions is not optimal.

3.6.1 Uniform Meshes

This example illustrates creating a multiple mesh model. To create the first mesh:

- On the Model menu, click Edit Meshes.
- Click New.
- In the Max X box, type 5.0, in the Max Y box, type 1.0, and in the Max Z box, type 1.0.
- In the X Cells box, type 50, in the Y Cells box, type 10, and in the Z Cells box, type 10.
- Click OK.

An example is shown in *Figure 4*.

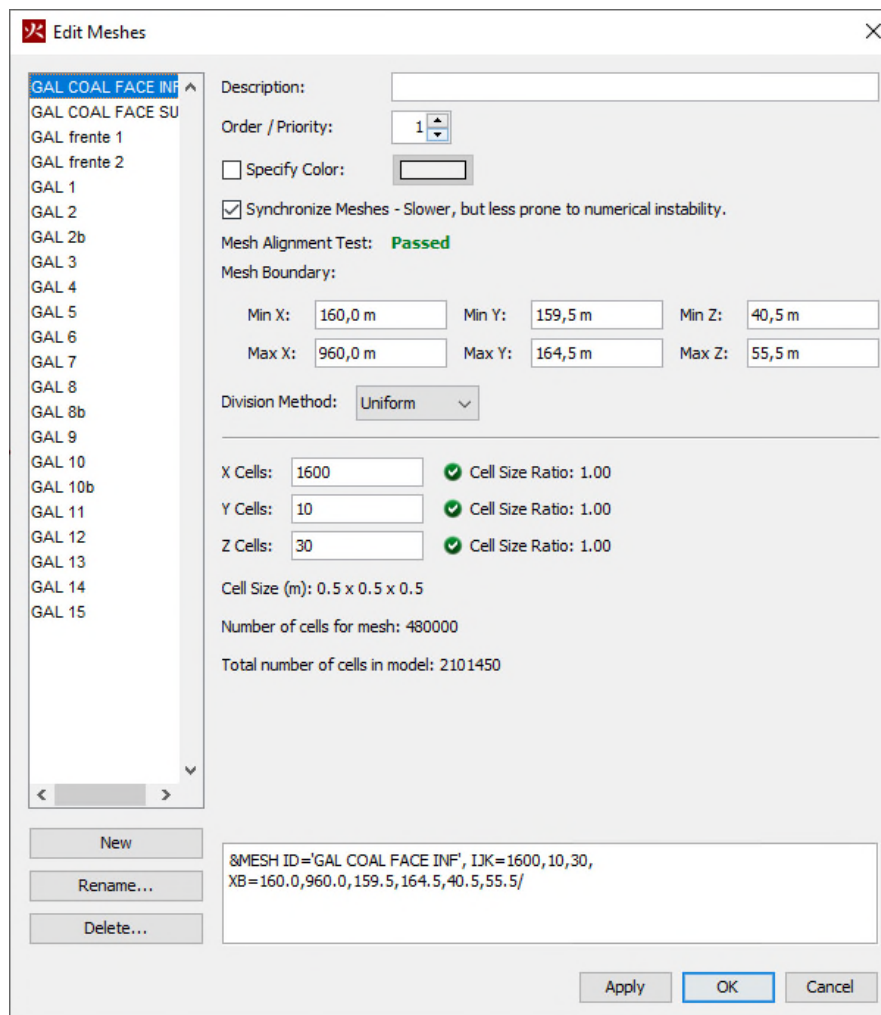


Figure 4 – Pyrosim Options to Edit Meshes

3.6.2 Non-uniform Meshes

To create a second, nonuniform mesh:

- On the Model menu, click Edit Meshes.
- Click New.
- In the Min X box, type 0.0, in the Min Y box, type 1.0, and in the Min Z box, type 0.0.
- In the Max X box, type 1.0, in the Max Y box, type 3.0, and in the Max Z box, type 1.0.
- In the Division method box, select Non-Uniform.
- In the table, enter the data shown in *Table 1*.
- Click OK.

Direction	Number of Cells	Size
X	10	0.1
Y	10	0.1
Y	5	0.2
Z	10	0.1

Table 1 – Example of non-Uniform Mesh Parameters

3.6.3 Multiple Meshes

The term “multiple meshes” means that the computational domain consists of more than one rectangular mesh, usually connected, although this is not required. In each mesh, the governing equations can be solved with a time step based on the flow speed within that particular mesh.

3.7 Materials

To simulate a surface made of heat-conducting solids or a fuel, you must specify a material that describes certain thermal properties and pyrolysis behaviour. PyroSim offers two categories of materials: solid materials and liquid fuels.

To create a new material, you can use the Edit Materials dialog. On the Model menu, click Edit Materials.

3.7.1 Solid Materials

Examples of solid materials include brick, gypsum board, and upholstery. To create a solid material:

- In the Edit Materials... dialog, click New.
- In the Material Name box, type the name of the new material.
- In the Material Type box, select Solid.
- Click OK.

The Thermal Properties tab provides the options shown in *Table 2*.

Parameter	Description
Density	The material's density.
Specific heat	The material's specific heat. Specific heat can be specified as a function of temperature.
Conductivity	The material's heat conductivity. Conductivity can be specified as a function of temperature.
Emissivity	The material's emissivity. A value of 1.0 corresponds to maximum radiation.
Absorption coefficient	This coefficient refers to the depth over which thermal radiation can be absorbed.

Table 2 – Thermal Properties

The By Reaction tab provides the options shown in *Table 3*.

Parameter	Description
Heat of reaction	Heat yield of this reaction. This must be a positive number.
Endothermic / exothermic	Specifies if the heat yield is endothermic or exothermic.
Residue	The material that will be used to represent the residue. If there is only one material defined in PyroSim, this option will not be available.

Table 3 – By-product Options

3.7.2 Liquid Fuels

Examples of liquid fuels include kerosene and ethanol. To create a liquid fuel:

- In the Edit Materials... dialog, click New.
- In the Material Name box, type the name of the new material.
- In the Material Type box, select Liquid Fuel.
- Click OK.

The thermal properties tab for liquid fuels is identical to the thermal properties tab for solid fuels.

3.8 Surfaces

Surfaces are used to define the properties of solid objects and vents in the FDS model. The surface can use previously defined materials in mixtures or layers. By default, all solid objects and vents are inert, with a temperature that is fixed at the ambient

temperature (set in the Simulation Parameters dialog). In addition to defining heat conduction in a solid, surfaces can also be used to define a burner, specify the ignition temperature for an object, give a vent a supply velocity, and set the many other properties supported by FDS.

3.8.1 Surface Creation

To create, modify, and delete surfaces, you can use the Edit Surfaces dialog. To open the surface manager dialog, on the Model menu, click Edit Surface Properties, as shown in *Figure 5*.

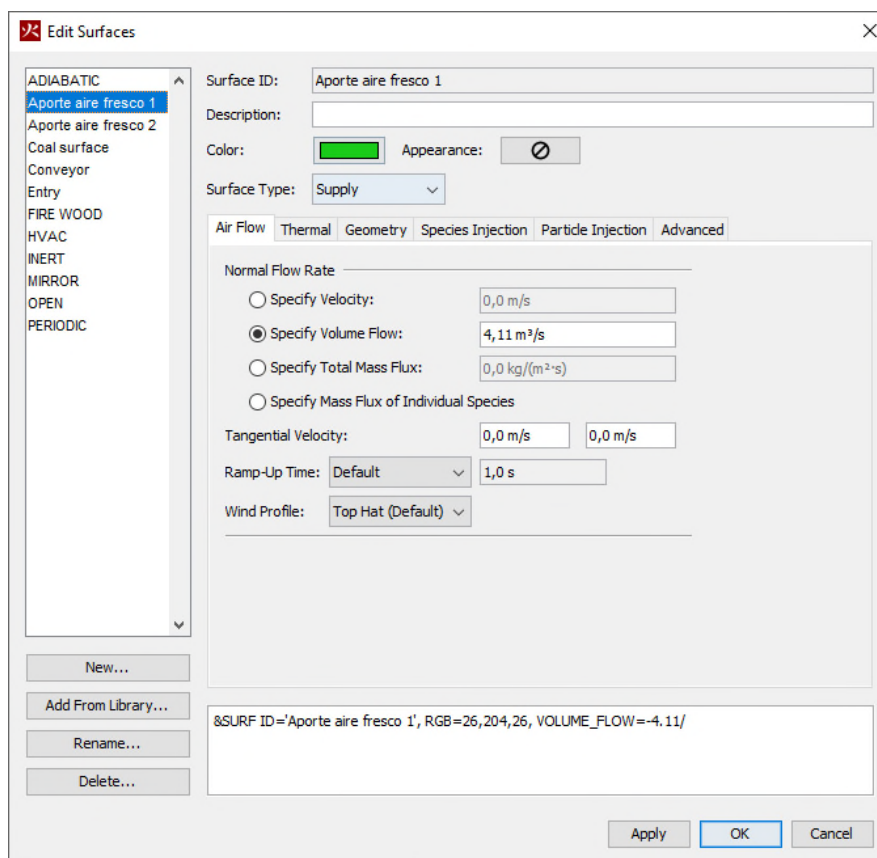


Figure 5 – Pyrosim Options to Edit Surfaces

3.8.2 Surface Types

PyroSim aids the user by organizing the surface options into logical types, such as a burner, to define a simple fire or a layered surface to represent a solid, heat conducting wall. The available surface types are described below.

Adiabatic

There is no net heat transfer (radiative or convective) from the gas to the solid.

Inert

This surface remains fixed at the ambient temperature. Heat transfer does occur from gases to INERT surfaces.

Open

This surface is used only for vents on the exterior mesh boundary. OPEN denotes a passive opening to the outside and is often used to model open doors and windows.

HVAC

This surface is used only for vents that are part of an HVAC system.

Burner

This surface type represents a fire with a known heat release rate or mass (fuel) loss rate.

Parameters for burner fires are arranged in two groups: heat release and particle injection. Heat release options are shown in *Table 4*.

Parameter	Description
Heat Release Rate (HRR)	The heat release rate per unit area of this burner.
Mass Loss Rate	The mass loss rate per unit area of this burner.
Ramp-Up Time	At the beginning of the simulation, this surface will not be burning. This field allows you to describe how the heat release ramps up from ambient to the specified value.
Extinguishing Coefficient	This parameter governs the suppression of the fire by water.
Surface Temperature	The surface temperature of this burner. The value TMPA represents ambient temperature.
Convective Heat Flux	The heat flux per unit area at the surface.
Net Heat Flux	The net heat flux per unit area at the surface.
Ramp-Up Time	This field allows you to describe how the temperature ramps up from ambient to the specified value.
Emissivity	This parameter controls how the surface radiates heat. Using a value of 1.0 makes this surface a black body. Lower values increase the amount of radiated heat.

Table 4 – Burner Options

Supply

This surface represents a vent that injects air into the simulation domain. The parameters for supply surfaces are arranged in four groups: air flow, temperature, species injection, and particle injection.

Exhaust

Exhaust surfaces can be used to remove gas from the simulation domain. The specification of their air movement parameters is identical to that of a supply surface, but instead of the velocity or flux driving air into the domain, they are pulling air out. Exhaust surfaces do not have the option to apply injection or geometry properties.

Layered

Layered surfaces are composed of one or more material definitions. Materials include solid and liquid substances such as concrete, pine, and ethanol.

Layered surfaces have five groups of options: material layers, surface props, reaction, species injection, and particle injection.

3.9 Geometry

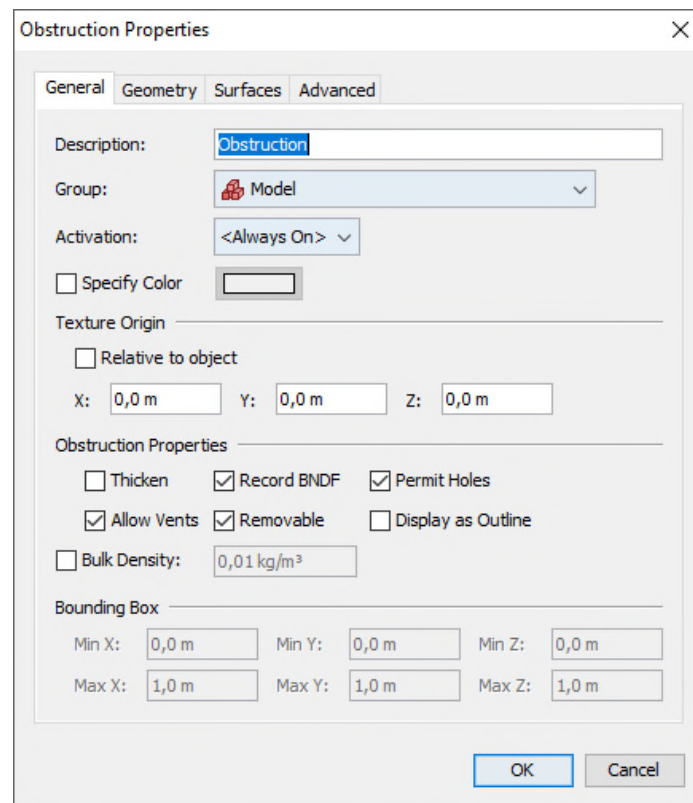
Geometry can either be created through dialogs or by using the drafting tools in the 2D or 3D views. There are typically three types of geometry that can be created in PyroSim:

- Obstructions – solid barriers to flow.
- Holes – negative regions that carve holes in obstructions.
- Vents – patches that define a sub region on an obstruction's surface that can have a different surface than the rest of the obstruction.

3.9.1 Obstructions

Obstructions are the fundamental geometric representation in FDS. In FDS, obstructions are rectangular, axis-aligned solids defined by two points. Surface properties are assigned to each face of the obstruction. In PyroSim, obstructions can take any shape, have any number of faces, and have different surfaces applied to each face.

To create a new obstruction, either use an obstruction drawing tool or on the Model menu, click New Obstruction.



The image shows a software dialog box titled "Obstruction Properties" with a close button (X) in the top right corner. It contains four tabs: "General", "Geometry", "Surfaces", and "Advanced". The "General" tab is selected and contains the following fields and options:

- Description:** A text field containing the word "Obstruction".
- Group:** A dropdown menu showing "Model" with a red cube icon.
- Activation:** A dropdown menu showing "<Always On>".
- Specify Color:** An unchecked checkbox next to a small color selection box.
- Texture Origin:** A section with an unchecked checkbox "Relative to object" and three input fields for X, Y, and Z coordinates, all set to "0,0 m".
- Obstruction Properties:** A section with several checkboxes:
 - ☐ Thicken
 - ☒ Record BNDF
 - ☒ Permit Holes
 - ☒ Allow Vents
 - ☒ Removable
 - ☐ Display as Outline
 - ☐ Bulk Density: 0,01 kg/m³
- Bounding Box:** A section with six input fields for minimum and maximum coordinates:
 - Min X: 0,0 m, Min Y: 0,0 m, Min Z: 0,0 m
 - Max X: 1,0 m, Max Y: 1,0 m, Max Z: 1,0 m

At the bottom right of the dialog are "OK" and "Cancel" buttons.

Figure 6 – Obstruction Properties

3.9.2 Vents

Vents have general usage in FDS to describe a 2D rectangular patch on the surface of a solid obstruction or on a mesh boundary. A vent may have a different surface applied to it than the rest of the obstruction to which it is attached.

Taken literally, a vent can be used to model components of the ventilation system in a building, like a diffuser or a return. In these cases, the vent coordinates form a plane on a solid surface forming the boundary of the duct.

Vents can either be drawn by opening the Model menu and clicking New Vent.

3.10 Reactions

The PyroSim interface supports only the single-step, mixing controlled combustion model to account for the evolution of the fuel gas from its surface of origin through the combustion process.

In FDS, there are two ways of designating a fire. The first is to specify a Heat Release Rate Per Unit Area (HRRPUA) as part of a surface, the other is to specify a HEAT_OF_REACTION, along with other thermal parameters, as part of a material. In both cases, the mixture fraction combustion model is used.

To edit a reaction:

- On the Model menu, click Edit Reactions.
- On the Fuel tab, you may enter either the number of atoms in the reaction equation or specify the reacting species. By default, PyroSim adds a custom species called REAC_FUEL to serve as the default fuel Species. If the species is not one defined by FDS, then the fuel composition must be defined as well. Note that not all of the implicitly defined species are reactive. The input for a polyurethane reaction described in the SFPE Handbook.
- On the Fire Suppression tab, you can enable fire suppression and then enter values for the Critical Flame Temperature (CRITICAL_FLAME_TEMPERATURE) and the Automatic Ignition Temperature (AUTO_IGNITION_TEMPERATURE).
- On the Byproducts tab, you can select either Specify release per unit mass oxygen (EPUMO2) or specify heat of combustion (HEAT_OF_COMBUSTION). You can also specify the CO yield, H2 yield, and Soot yield.

This page is
intentionally
left blank.

4 PRACTICAL CASE TO BUILD A MINE MODEL FOR CFD FIRE SIMULATIONS

Before modelling a mine fire with the FDS software, it is important to understand how to choose the fire features.

4.1 Introduction

The CFD software FDS (Fire Dynamic Simulator) is employed in this study and several practical cases are considered to better illustrate the procedure.

The steps to model a fire in a coal mine are the following ones:

4.1.1 Choice of the Mine Risk Scenario

The mine geometry strongly conditions the choice of scenario, since it is usually considered the worst case. This is dependent on some mine features, such as the slope, the cross-sectional area or the ventilation conditions during mine exploitation.

4.1.2 Modelling the Mine Geometry

It is possible to model the entire mine, but this is something that is usually avoided, since the computational cost is quite elevated. For studies involving the whole mine, the 1D software is more commonly used, while the CFD simulations are more intended for detailed studies.

4.1.3 Modelling the Fire

The fire features depend on several aspects, such as the burning material, the oxygen conditions or the presence of some gases in the atmosphere. It is important to have a look at the data that shows the features of the fires that have been registered in coal mines to make the right choice.

4.1.4 Assessing the Results

The assessment focuses on several variables: temperature, CO concentration, visibility, because all of them have an impact on miners who try to escape from the fire. Besides, it is important to know the highest temperatures registered in the mine, since this affects the features that devices must have.

4.2 Input Parameters to Define the Mine Risk Scenario

A fire scenario can be defined by several aspects, such as:

- Burning material,
- Accident that has led to a fire,
- Fire location,
- Ventilation conditions,
- Emergency procedure.

In order to model correctly a fire in a coal mine, it is important to know well the four aspects previously listed.

4.2.1 Accidents with Highest Likelihood

According to NIOSH, the origin of the fires that have occurred in the USA are the ones shown in *Figure 7*.

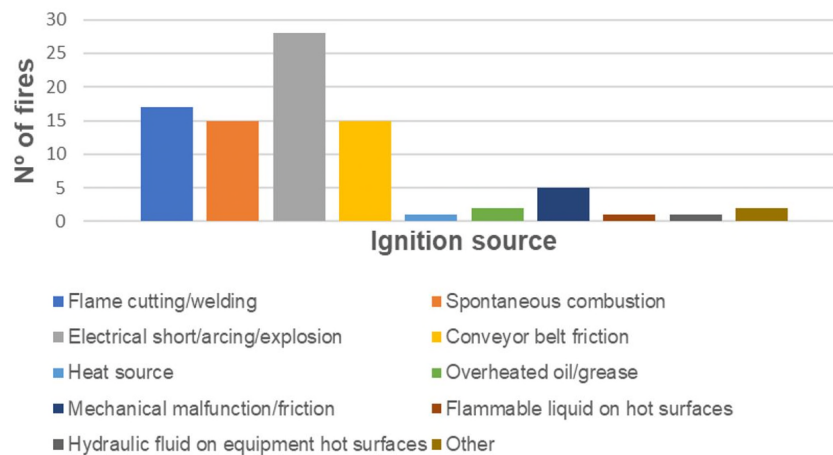


Figure 7 – Fires in US Coal Mines, 1990-1999, by Ignition Source (Source: NIOSH)

As seen in *Figure 7*, the most likely cause of a fire in a mine is an electrical short or arcing, followed by flame welding, conveyor belt friction and spontaneous combustion.

This allows fires that are most likely to take place in a coal mine to be better modelled.

4.2.2 Materials with Highest Likelihood of Ignition

Also, we can classify the fires in coal mines according to the material in which the fire originates. According to the records of fires depending on the burning material, the classification is the one shown in *Figure 8*.

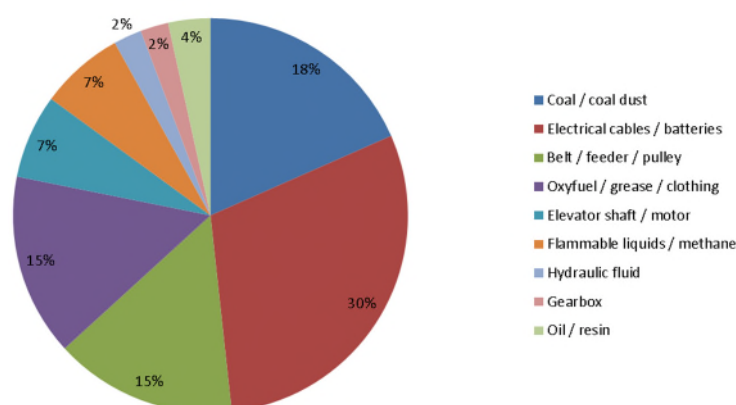


Figure 8 – Fires in US Coal Mines, 1990-1999, by Burning Material (Source: NIOSH)

Electrical equipment is the most likely material to burn in the case of a fire in a mine, followed by coal dust and belts.

4.2.3 Fire Location

The fire location is a key parameter, since the consequences can differ significantly, depending on this parameter. For example, in a coal mine containing regions with slope, the buoyancy effect will push the smoke towards the positive slope, and this will establish the smoke trajectory in some cases.

This can be appreciated in from *Figure 9*, where a specific region of a coal mine is represented and there are some sections with a strong slope.

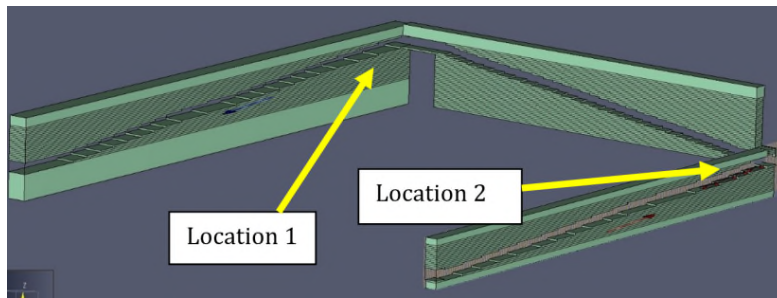


Figure 9 – Fire Location in a Mine

If a fire is placed in Location 2, the smoke will advance towards Location 1, because of the buoyancy effect, since the smoke density is much lower than the ambient density.

On the contrary, if a fire is placed in Location 1, it is not very clear which path the smoke will take, since in both senses the smoke will find a negative slope.

4.2.4 Ventilation Conditions

The ventilation conditions at the moment when a fire is declared is of great importance, since it will have a big influence in the smoke trajectory.

In all operative fronts there has to be a permanent ventilation to evacuate the coal in the atmosphere, as shown in Figure 10.

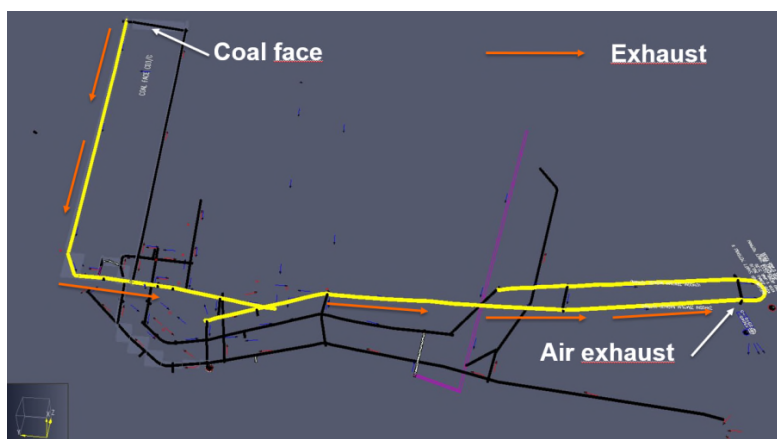


Figure 10 – Ventilation Manoeuvre During Exploitation Phase

These conditions need to be specified in the software to model correctly the fire's evolution across the simulation's duration.

4.2.5 Emergency Procedure

The emergency procedure needs to be specified as well, since some actions will take place after a fire is declared and they will surely affect the ventilation conditions.

The ventilation maneuver for an emergency consists of pushing the smoke on a specified direction, as shown in *Figure 11* and *Figure 12*.

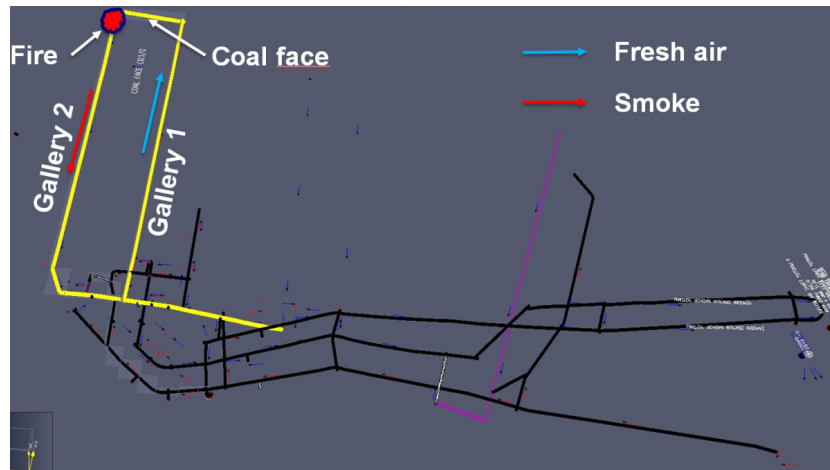


Figure 11 – Ventilation Maneuver during Emergency Phase (I)



Figure 12 – Ventilation Maneuver during Emergency Phase (II)

It is important to know the ventilation maneuver, since it needs to be included in the fire model.

4.3 CFD Model

This FDS software solves the Navier Stokes equations and the equations of turbulence and calculates the evolution of the main variables related to the fire (visibility, temperature, CO concentration, extinction factor, etc).

The steps to model a fire in a coal mine are the ones explained in this section.

4.3.1 Creation of the Mine's Geometry

The first feature to be defined is the mine geometry.

Normally, the mine's geometry is available in a cad design model, as shown in *Figure 13* for a specific coal mine.

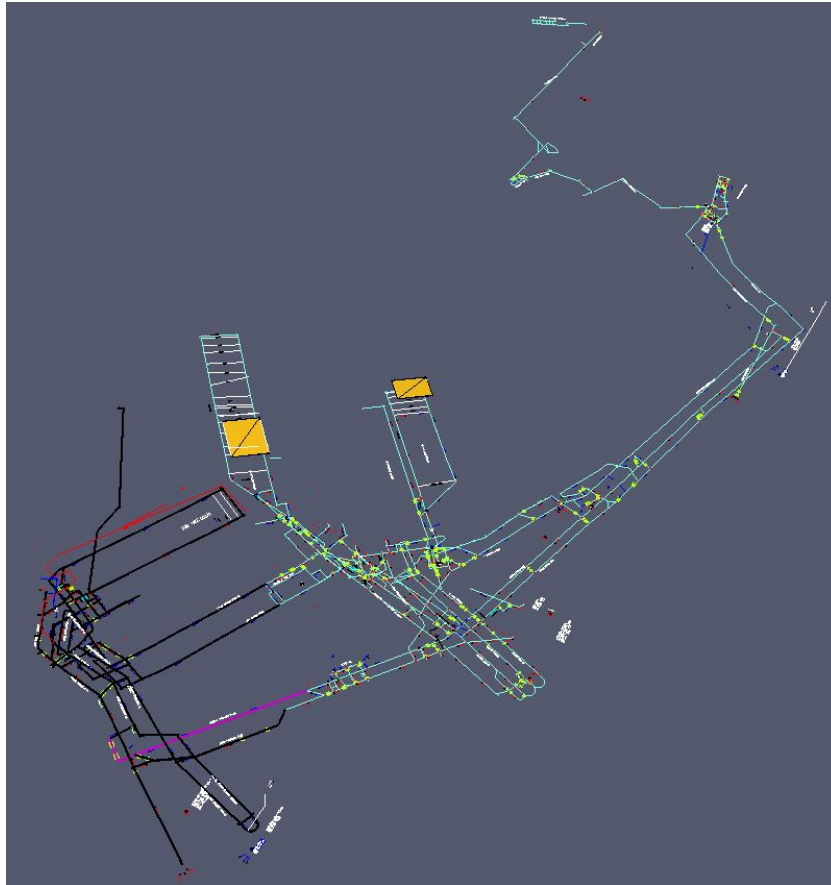


Figure 13 – View of the Mine Geometry

Besides, it is necessary to know the cross-sectional area, the perimeter, and the presence of obstacles to better model the smoke behavior, as shown in *Figure 14*.



Figure 14 – View of the Mine Section

It is possible to work with the whole mine or just a region of it, if we consider that it is enough for our goal, as shown in *Figure 15 – Figure 18*.

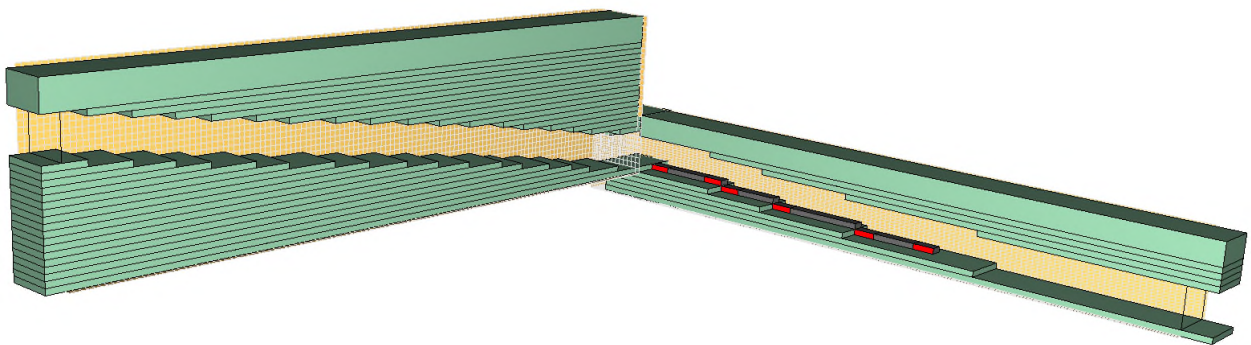


Figure 15 – Model Focused on a Small Mine Region

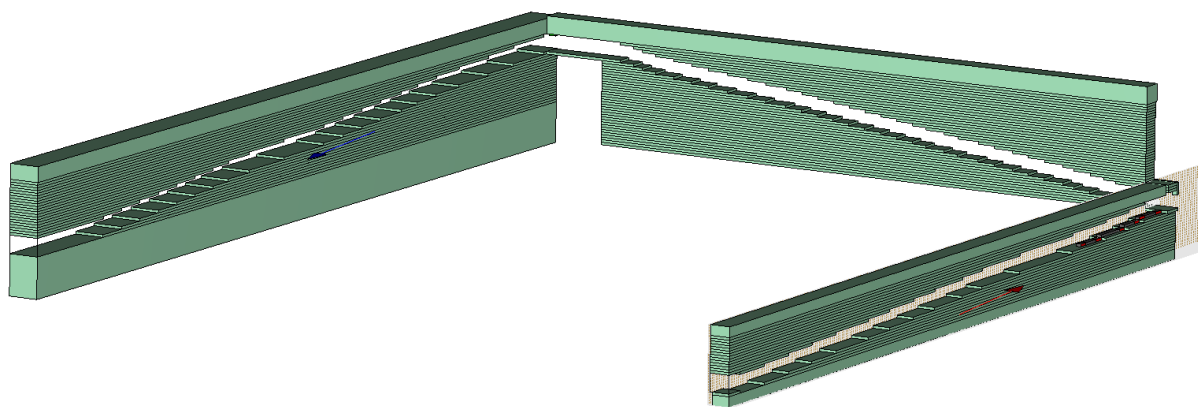


Figure 16 – Model Focused on a Larger Mine Region

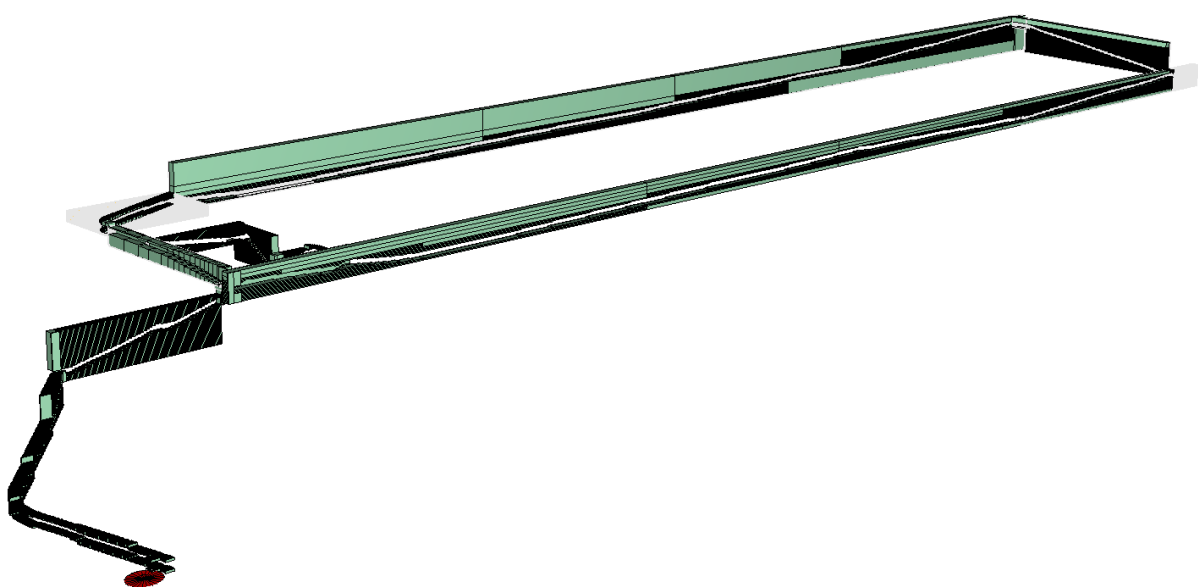


Figure 17 – Model Focused on an Even Larger Mine Region

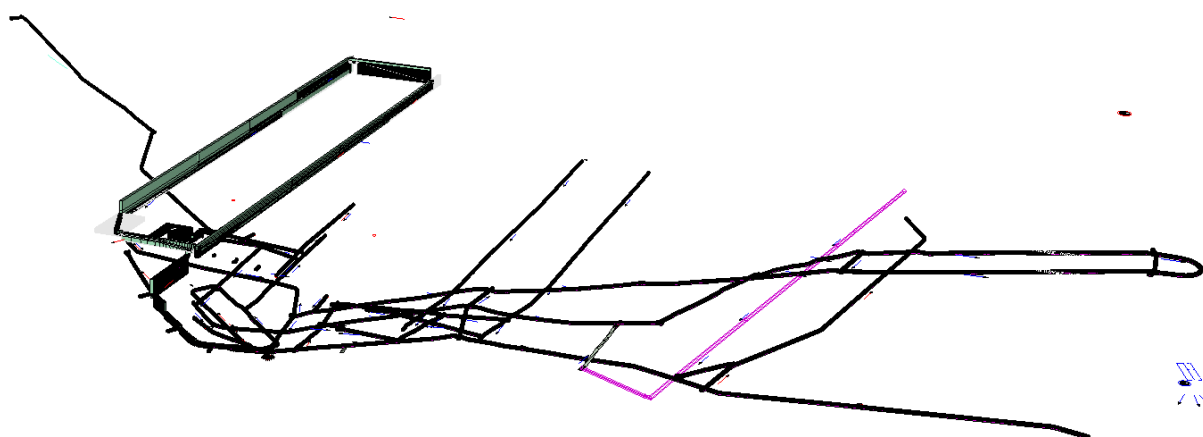


Figure 18 – Model Containing the Whole Mine

Since the CFD simulations have a high computational cost, these simulations mandatorily have to be closely related to 1D simulations.

The 1D simulations contain the whole mine and it is possible to conduct as many 1D simulations as one may desire, since their computational cost is very low.

Therefore, when it is necessary to study the evolution of the fire in detail, the steps to take are as follows:

- Carry out several 1D calculations.
- Obtain the fire and the velocity evolution across the whole mine throughout time.
- Choose the region where one wants to launch the CFD simulation.
- Impose the velocity field as boundary conditions in the 3D model.

Mesh

A mesh has to be defined for the model and the mesh cells in the fire vicinity have to comply with the following formula, which establishes the maximum edge dimension.

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

4.3.2 Fire Definition

The next step consists of defining the fire that will be chosen.

Chemical Reaction

Several fires can be considered, depending on the material that is burning. It is possible to establish the chemical reaction in the model, as shown in *Figure 19*.

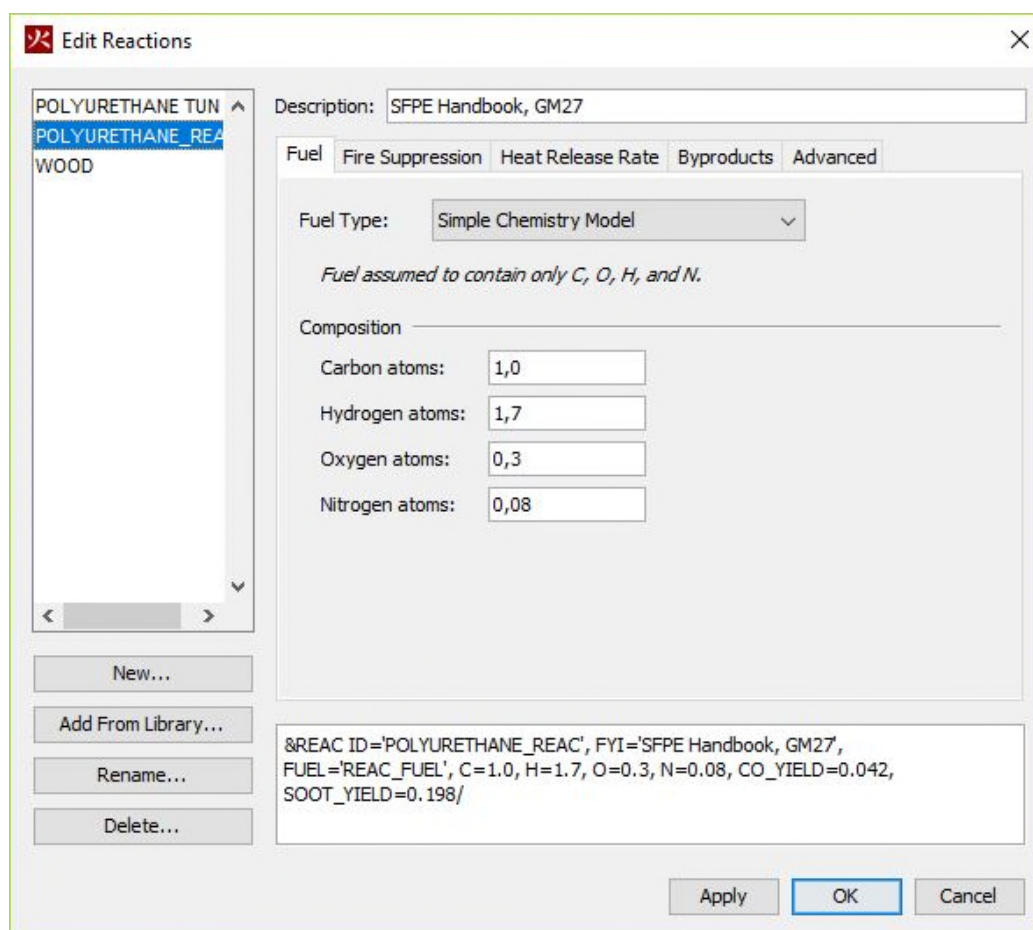


Figure 19 – Edit Reactions Window

Burning Surface

Besides, it is necessary to know the surface that is burning, something that can be determined thanks to the data provided by NIOSH that has already been presented.

For example, if it is considered that a conveyor belt is burning, it is necessary to choose a surface with a coherent value, according to the data from NIOSH. Besides, this surface has to be defined as a Fire Surface, where the chemical reaction is defined before considering where the HRR (Heat Release Rate) can be specified.

HRR Curve

Each fire that one may consider for the CFD simulation has to be defined by an HRR curve.

In order to know the HRR curve, it is possible, as well, to consider the information from fire tests already done. For example, in the case of a conveyor belt, there are six fire tests that have been effectuated, as outlined *Table 5*.

Type of Conveyor Belt	HRR Peak (MW)	Heat Released (MJ)
Type-1	7	9570
Type-2	6.2	8768
Type-3	5.2	6436
Type-4	4.8	6642
Type-5	4.2	6030
Type-6	3.8	6337

Table 5 – Results of Fire Test with Conveyor Belts

Each test presents an HRR curve that has been measured during the test. For example, for the Type 1 conveyor, the HRR curve is shown in *Figure 20*.

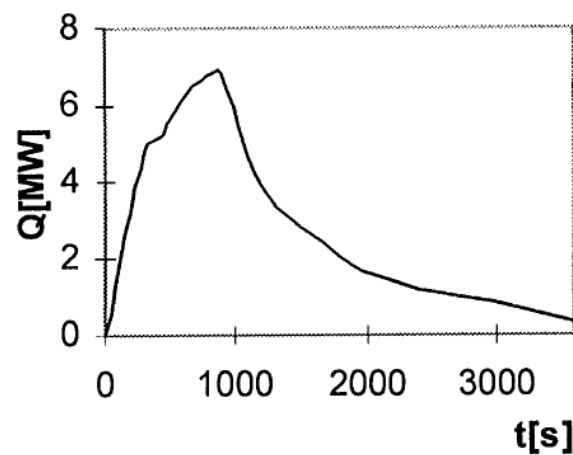


Figure 20 – Theoretical HRR Curve for Type 1 Conveyor Belt

A modelling of this curve may be the one shown in *Figure 21*.

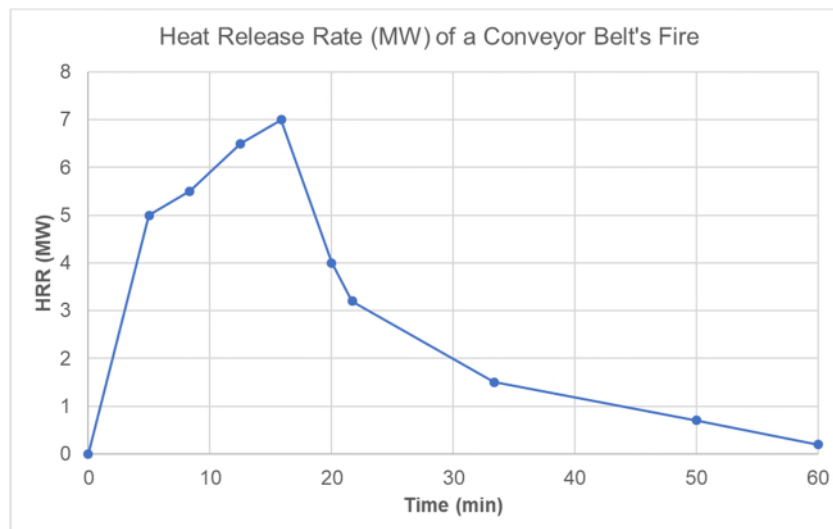


Figure 21 – Modelled HRR Curve for Type 1 Conveyor Belt

There is a window in the CFD model to specify the HRR curve.

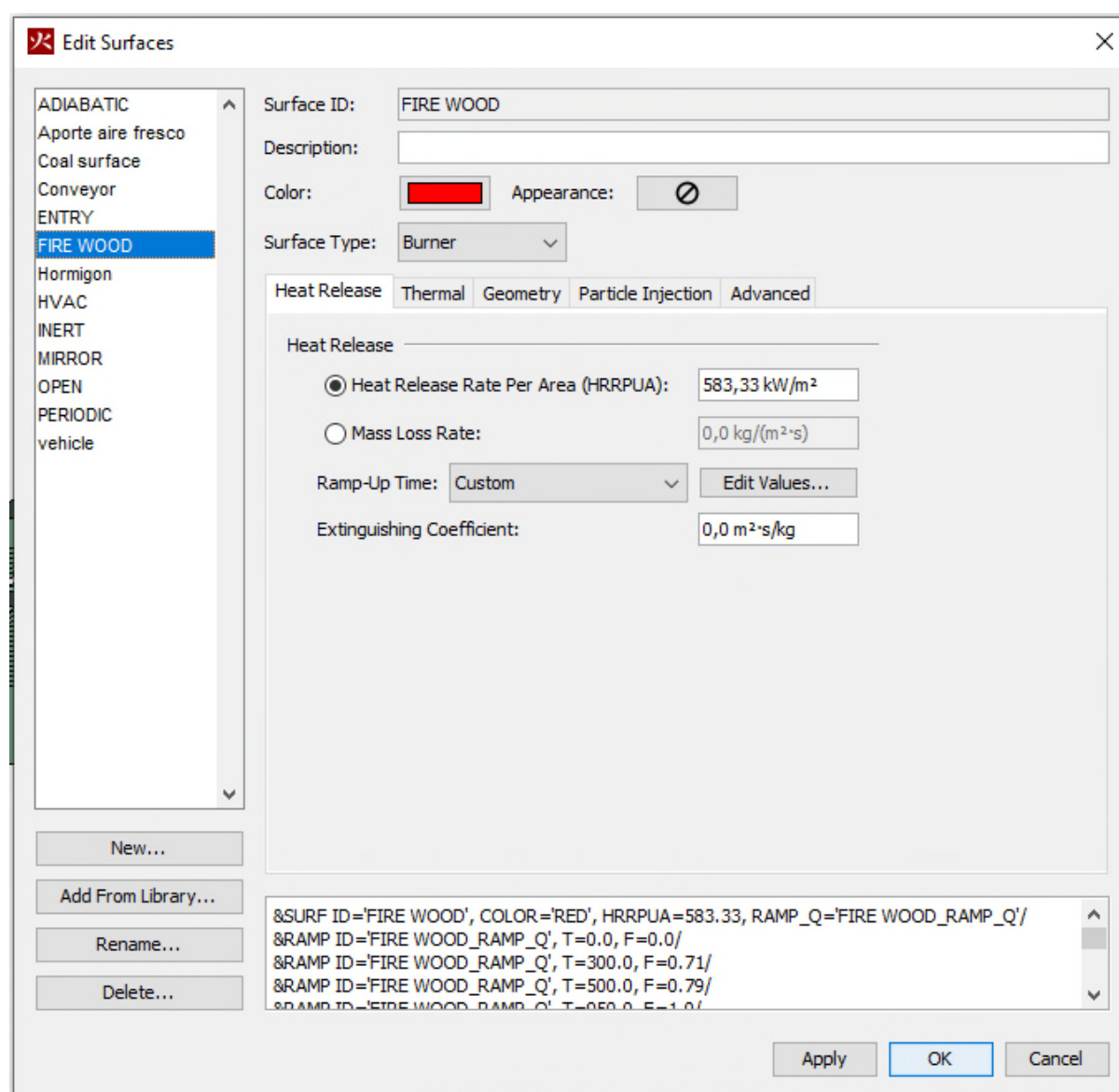


Figure 22 – HRR Curve Definition

Fire Location

The fire location has to be chosen to have all the model defined. Usually the location chosen is one that is demanding in terms of safety, since the end goal of the simulation is to make an assessment on the mine's safety.

4.3.3 Boundary Conditions

When a fire takes place, the ventilation regime that reigns at that moment will establish the smoke path. This feature has a great importance, since the effect on workers will depend very much on the initial path that the smoke takes.

The following example is quite self-explanatory on how the initial ventilation regime conditions the workers' safety.

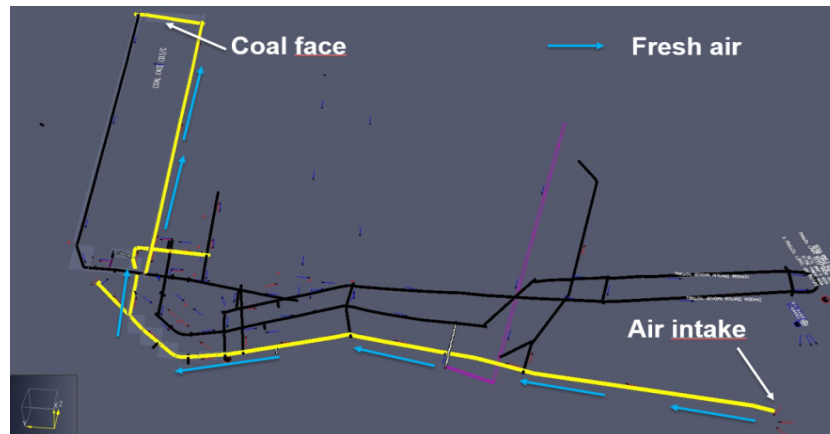


Figure 23 – Normal Operation in a Coal Mine (I)

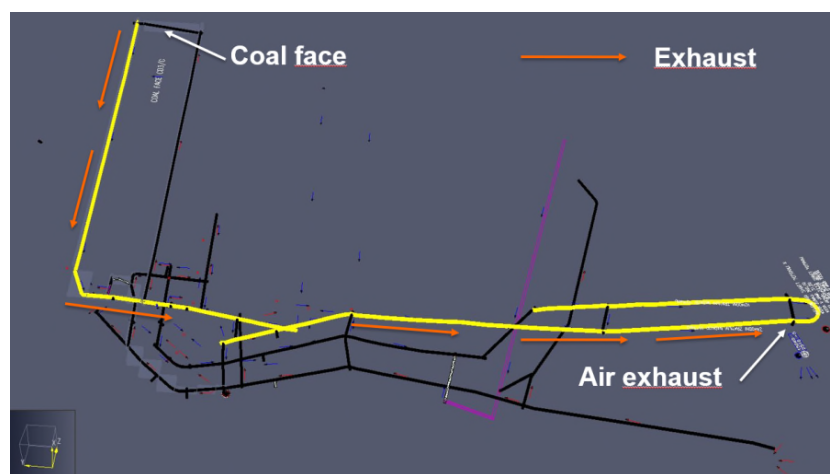


Figure 24 – Normal Operation in a Coal Mine (II)

In addition, a good ventilation will establish a lower CO generation at the fire source, which means that this aspect unveils more important aspects.

The boundary conditions that are employed usually are:

- Velocity inlet: for the entrance, where an airflow is specified.
- Pressure outlet: for the outlet of the model.

The velocity field imposed in the inlet is obtained thanks to the 1D simulations of the whole mine, where all the working conditions have been introduced.

The CFD model has a window where the velocity inlet can be specified.

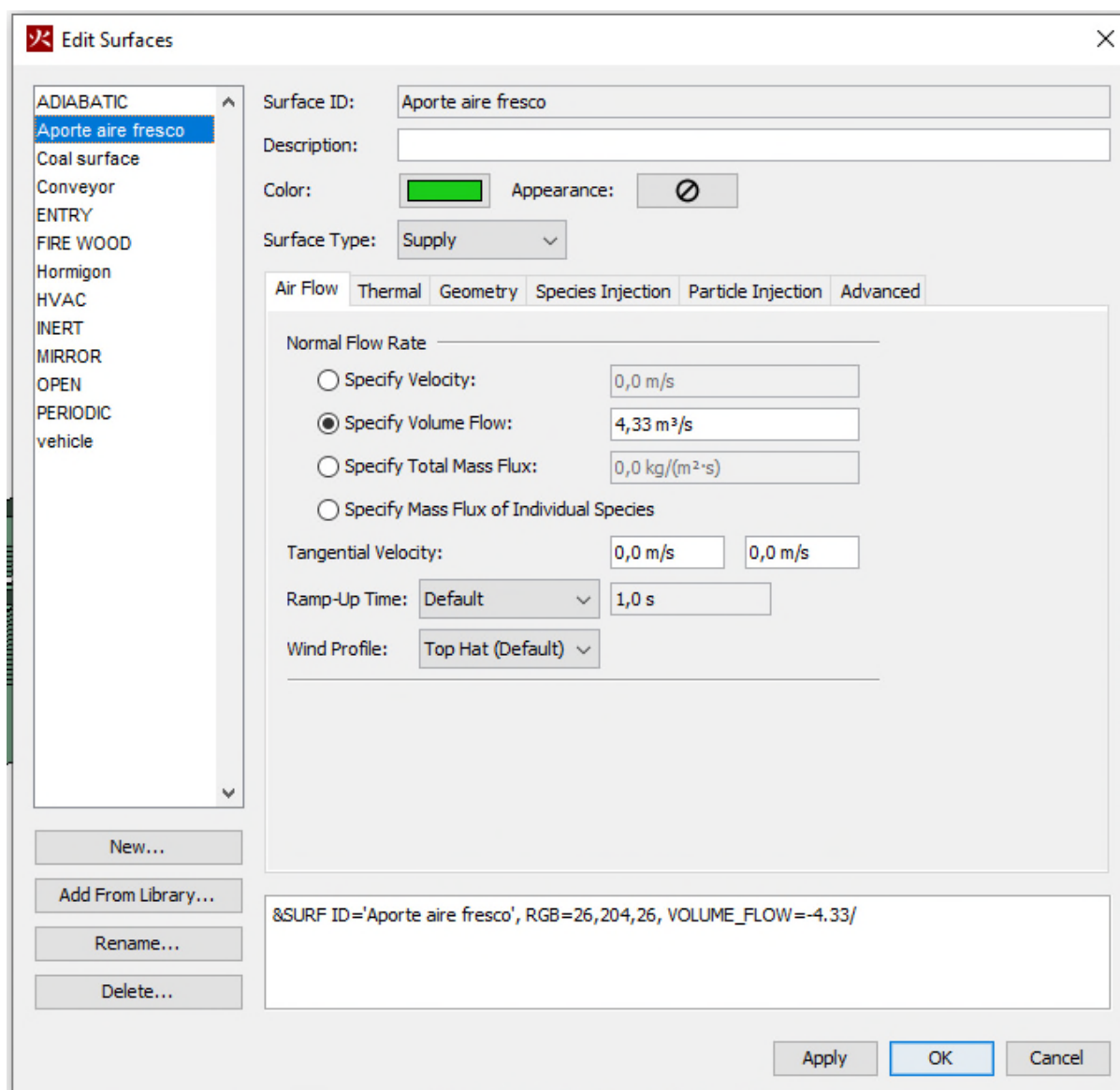


Figure 25 – Boundary Conditions Definition

The velocity field is obtained through 1D simulations with a software such as Ventsim, as shown in Figure 26 and Figure 27.

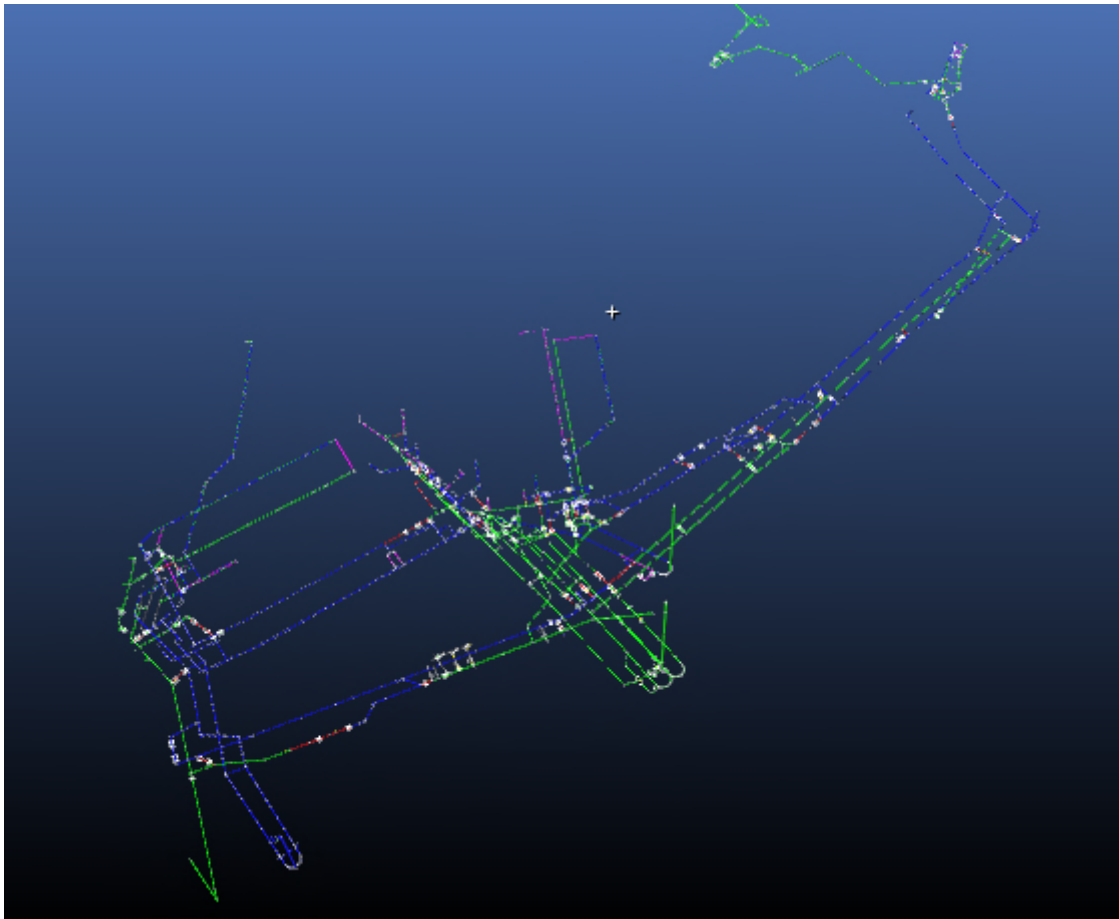


Figure 26 – Mine Model with Ventsim (I)

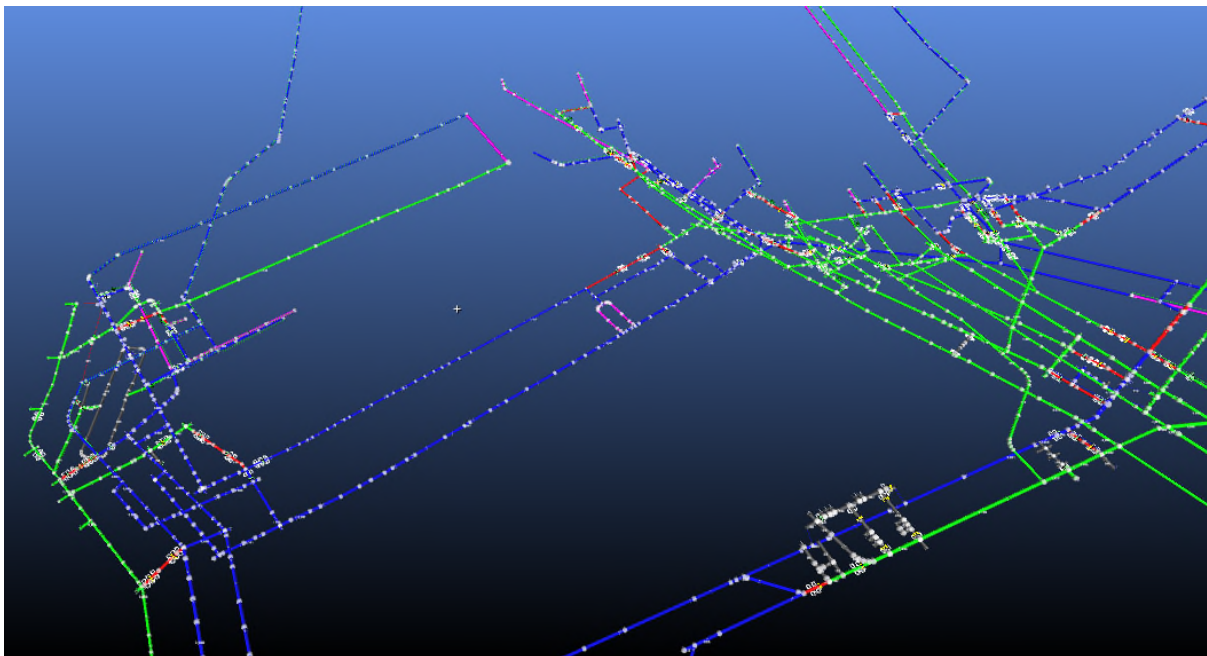


Figure 27 – Mine Model with Ventsim (II)

This software allows airflows to be imposed in the model, as shown in *Figure 28*.

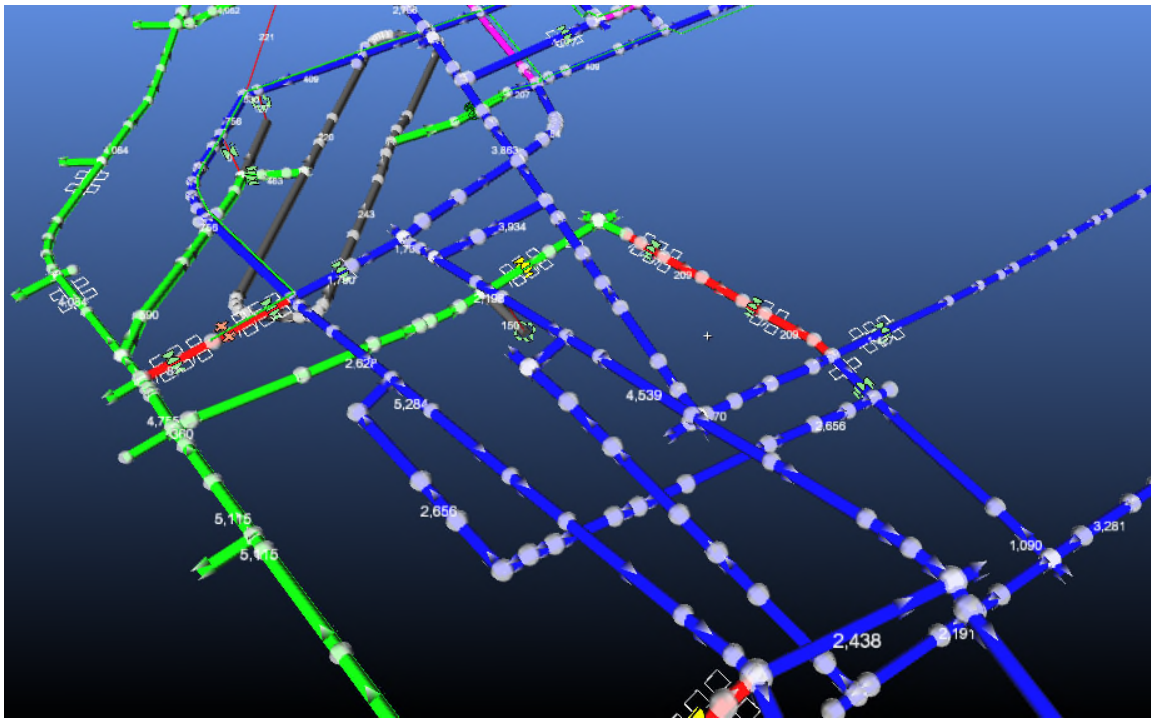


Figure 28 – Mine Model with Ventsim (III)

4.3.4 Emergency Procedure

The emergency procedure has several implications, both in terms of the fire evolution and of the fire service's intervention.

The main implication of the procedure is related to the ventilation regime evolution, because the goal of the emergency ventilation maneuver is to isolate the smoke, protect the miners and eventually allow an emergency intervention.

An example of the emergency ventilation maneuver in a coal mine is shown in *Figure 29* and *Figure 30*.

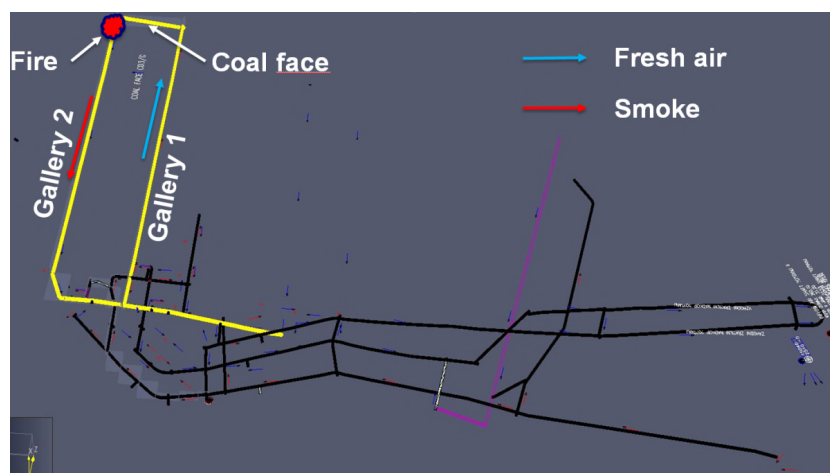


Figure 29 – Emergency Ventilation Regime (I)

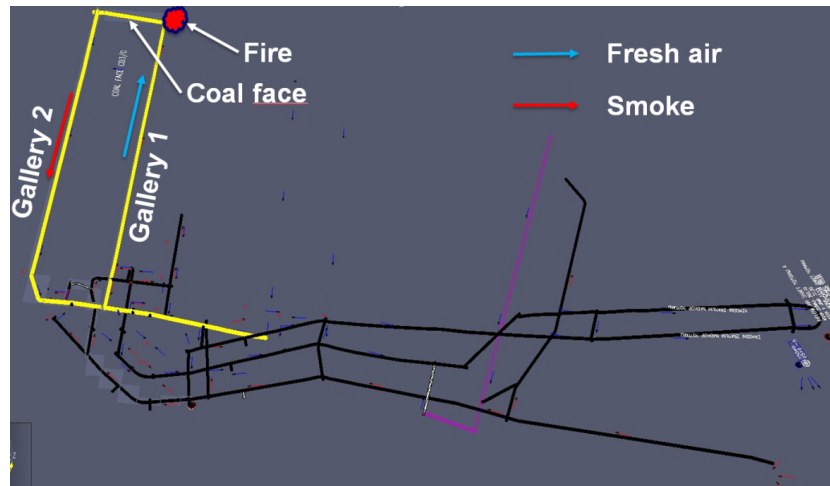


Figure 30 – Emergency Ventilation Regime (II)

The ventilation regime is established in the same window that was shown before and the data is obtained thanks to a 1D simulation in the mine.

These values can be calculated with a 1D software such as Ventsim, which is designed specifically for mines.

4.4 Results

Once the simulations are concluded, it is possible to obtain further detail about the main variables related to the fire that one may consider as important to make an assessment: visibility, CO concentration, velocity.

4.4.1 Results Measurement Points

An example of the values (temperature, CO concentration and visibility) obtained in several measuring points in the fire's vicinity are shown in *Figure 31 – Figure 33*.

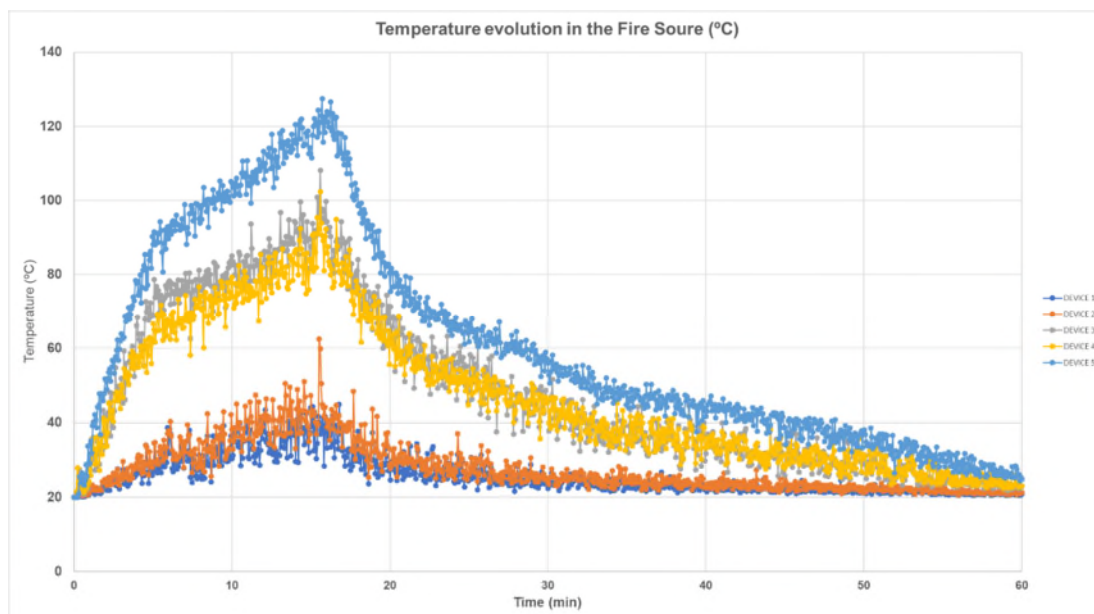


Figure 31 – Temperature Levels in Fire Vicinity

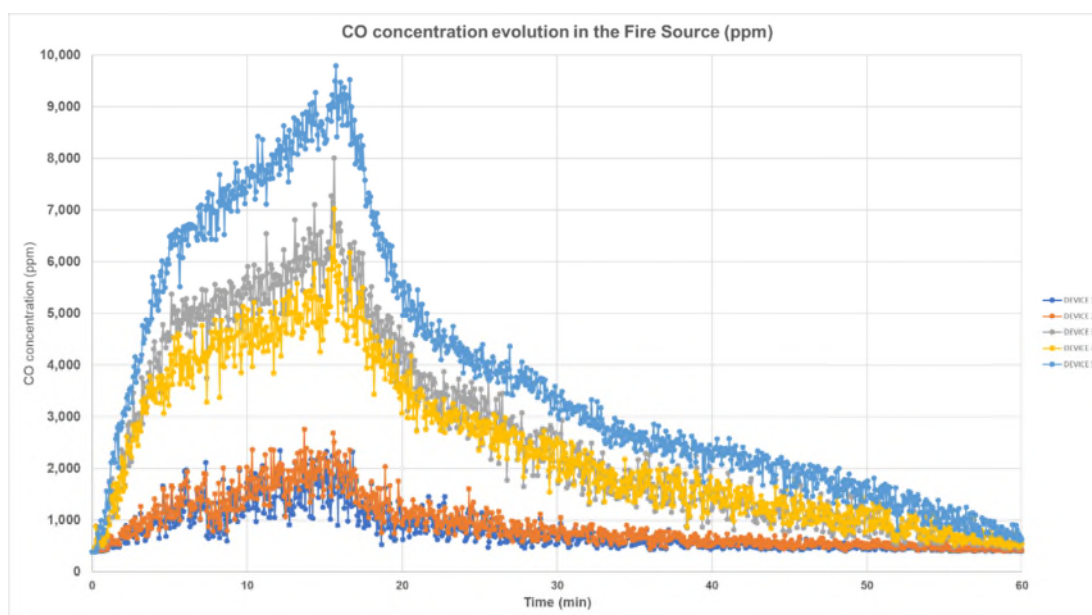


Figure 32 – CO Concentration Levels in Fire Vicinity

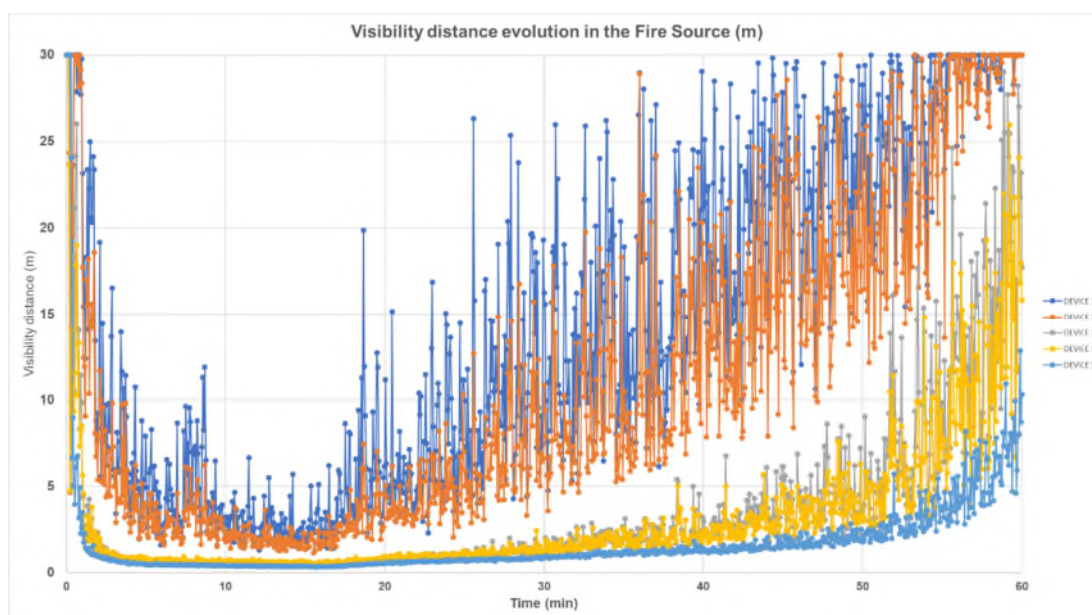


Figure 33 – Visibility Levels in Fire Vicinity

Also, it is possible to measure the evolution of these variables in the downstream region, which is important, since it allows to have an idea of the impact on the miners' safety, as shown in *Figure 34 – Figure 36*.

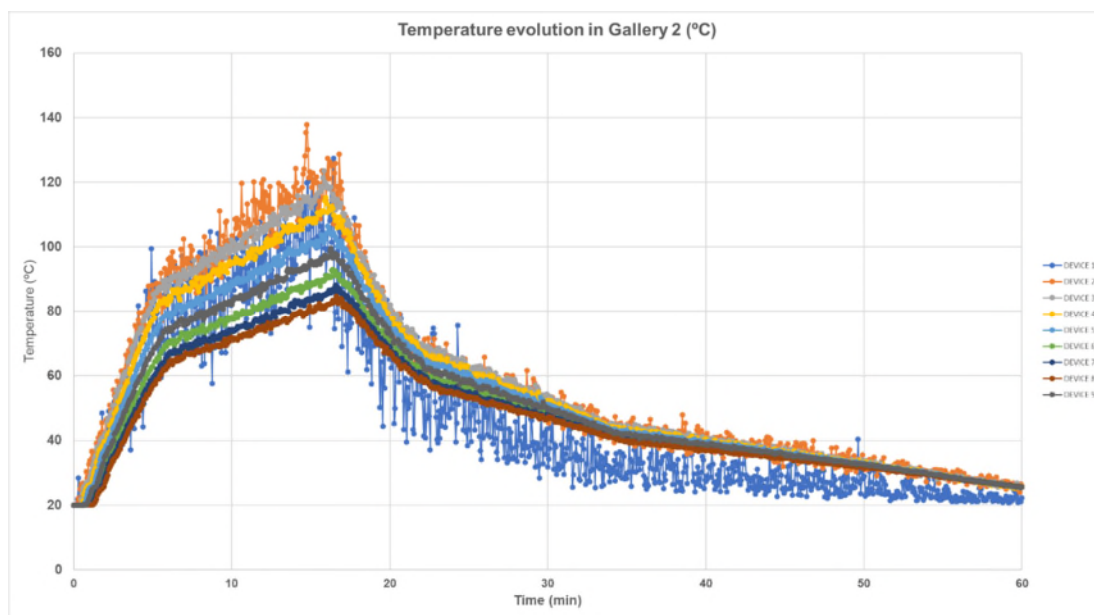


Figure 34 – Temperature Levels in Downstream Region

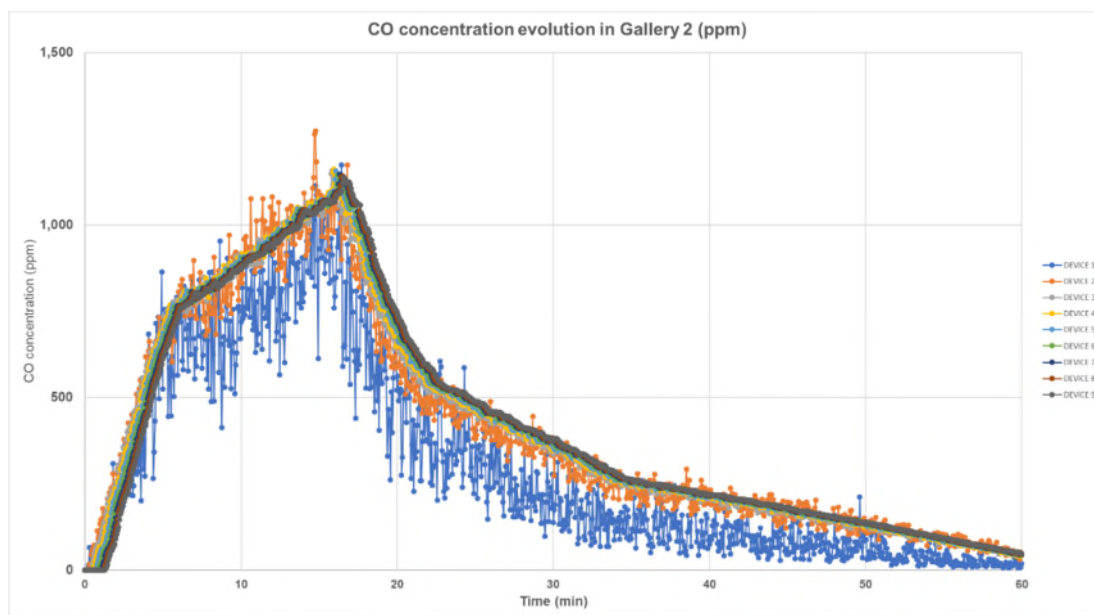


Figure 35 – CO Concentration Levels in Downstream Region

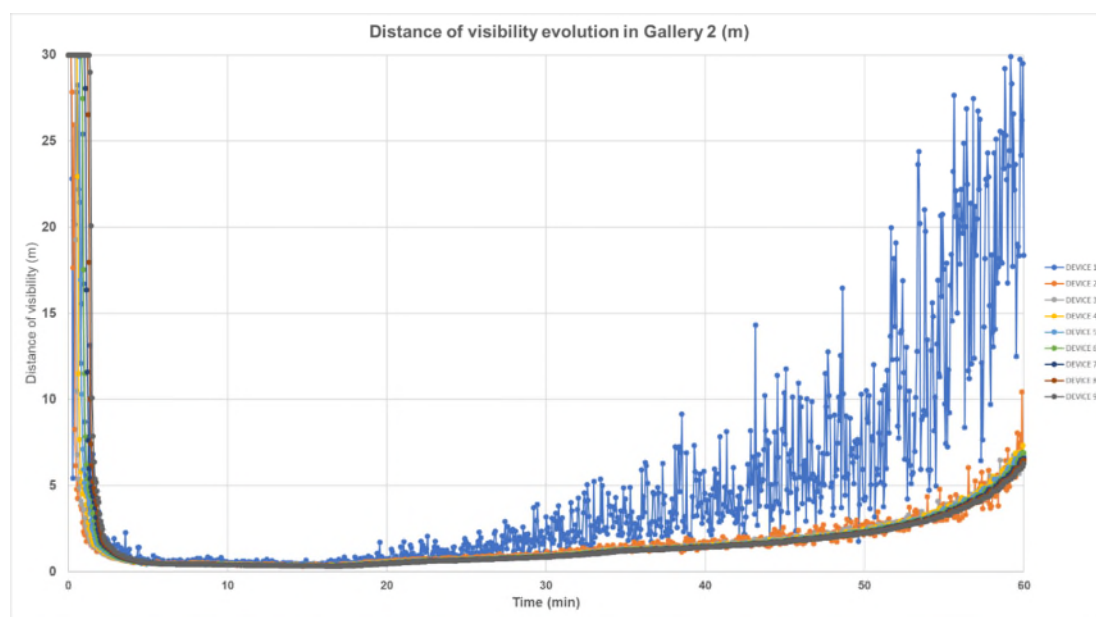


Figure 36 – Visibility Levels in Downstream Region

4.4.2 Results in a Plane of Measure

The evolution of these variables can be analyzed in a plane of measure, as shown in *Figure 37*.

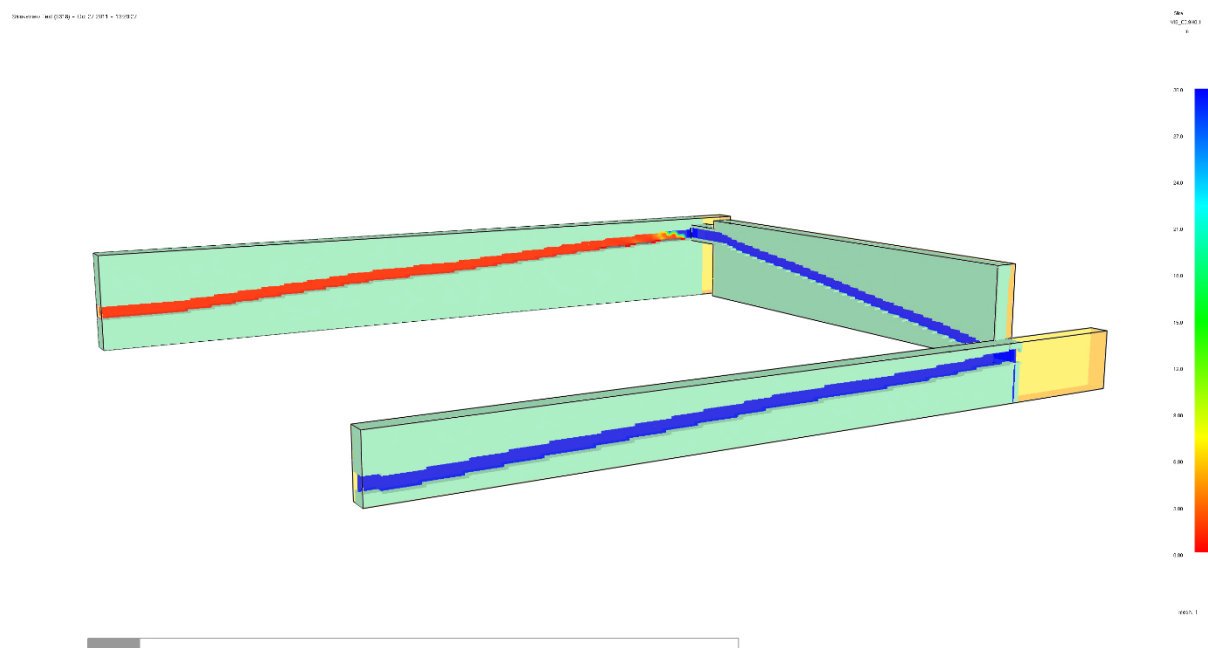


Figure 37 – Visibility Levels in the Whole Model

**This page is
intentionally
left blank.**

5 FURTHER INFORMATION

Geocontrol has experience in the domain of fire simulations in coal mines and is always looking forward to collaborating in projects related to coal mines. Further information about CFD fire simulations and studies carried out by Geocontrol can be solicited.

Please contact Rafael Sanchez at rsanchez@geocontrol.es

