INERIS

INSTITUT NATIONAL DE L'ENVIRONNEMENT INDUSTRIEL ET DES RISQUES

Perfecting a Tool for Simulating Accident Situations in Underground Networks

B C R D
Final Summary Report

Program authorization 1997 –
Subsidy No. 97-028 notified on June 27, 1997
(modification decision dated August 21, 2000)

Accidental Hazards Department

June 2001

Perfecting a Tool for Simulating Accident Situations in Underground Networks

B C R D
Final Summary Report

Program authorization 1997 –
Subsidy No. 97-028 notified on June 27, 1997
(modification decision dated August 21, 2000)

JUNE 2001
THE FOLLOWING PERSONS TOOK PART IN THE STUDY
E. RUFFIN, S. DAERON, L. FOURNIER, E. DANIN

	Written by	**Checked by**	**Approved by**
Name	E. RUFFIN	Y. MOUILLEAU	B. FAUCHER
Position	Programme Manager	Head of Unit	Operating Director
Signature	*Signed*	*Signed*	*Signed*

TABLE OF CONTENTS

ABSTRACT

For many years now, INERIS, the French National Institute for Industrial Environments and Hazards, has been a major player in safety at industrial installations.

Building on this experience, the institute has developed this research project, especially focusing its effort on the ability to use contemporary computer system advances in order to open up a new way to precisely simulate accidental events (excluding explosions) in confined locations. What is at stake is clearly safety in underground areas [1] and especially in tunnels, structures which are becoming more and more complex.

This project, which covers some of the requirements relating to the elaboration or assessment of Specific Danger Evaluations for road tunnels (as part of a new regulatory context in France), takes aim at the following primary goals:
- improving the evaluation of accidental effects in complex underground structures,
- drawing up a computer coupling for a single-dimensional model for networked ventilation systems along with a three-dimensional model for dispersion calculations (CFD model) covering smoke from fires and toxic emissions.
- performing experimental test cases in order to obtain data for validating the digital development work undertaken,
- validating the development work undertaken.

Among these goals, the computer coupling of the codes used for various kinds of calculations is clearly the key to an improved evaluation of accidental effects in confined locations. These effects are due to the change in behavior by the works (with a more or less complex ventilation system) due to the presence of the accidental source and the impact of the change in behavior of the works on the accidental source itself.

This project, therefore saw the implementation of development work based around the numerical coupling of two kinds of calculation:
- calculating ventilation networks with the aim of forecasting the overall behavior of a construction,
- calculating fluid dynamics as represented by the Navier-Stokes equations (CFD code) aimed at correctly taking into account the local behavior of an accidental source.

To implement this numerical coupling and its validation, the following have been created:

On the one hand, a new code for calculation ventilation networks – New-Vendis – with a layout that is suitable for exchanging data with and outside calculation code (CFD or other) and which allows for the numerous accidental and natural effects that are specific to a ventilation network,
- On the other hand, an experimental installation dedicated to obtaining validation data and more generally for the small scale simulation of accidental effects.

Lastly, the experimental data obtained is useful for validating numerical developments, but also in their own right, represent the first interesting results in relation to the small scale simulation of fires in confined locations.

1. INTRODUCTION

On the strength of its experience in the safety field, INERIS took an interest in improving the evaluation of the effects of accidents in underground areas.

In light of the context which saw the project initiated (in 1997), it seemed important to improve the resources available for simulating the effects of accidents in underground structures, and this for the following main reasons:

- it is important to be able to better forecast the operation and the sizing of the ventilation installations and of the various safety components that make it possible to avoid the effects of accident related phenomena - something that partially at least, conditions the cost of the structures, as well as being able to define the actions to be undertaken in order to limit the effects of accidents,

- the techniques used for forecasting the effects of fire and accidental gaseous emissions only apply for underground networks and in single tube tunnels, based on simplified limit conditions which are often hardly representative of reality,

- the effects of accidents in complex structures (underground transportation networks, major rail or road tunnels, underground buildings or storage facilities…) are practically impossible to predict, simultaneously, and with the necessary degree of precision, for both the overall behavior of the structure and any accidental local effects.

To properly understand the problem, let us take the example of fire.

Simulating a fire in an underground facility raises a number of physical phenomena (some of which are still not fully understood) such as of course the heating up of the air flow present in the tube where the fire is located as well as the production of smoke. Both of these basic characteristics will be modulated depending on whether the draught through the tunnel is a strong or a weak one and whether or not the smoke forms a hot layer (becoming stratified) or not or whether the smoke propagation entails backlayering or not.

At this stage, if one adds the fact that fire, given its heating power, may cause a variation in the balance of air flows within a ventilation network (heat draw), it becomes easy to understand that simulating only the fire phenomena in an underground structure cannot be achieved from just one kind of calculation code. At the very least, it requires taking into account **overall phenomena** (variations in air flows in the network) and taking into account **local phenomena** (three-dimensional smoke dispersion).
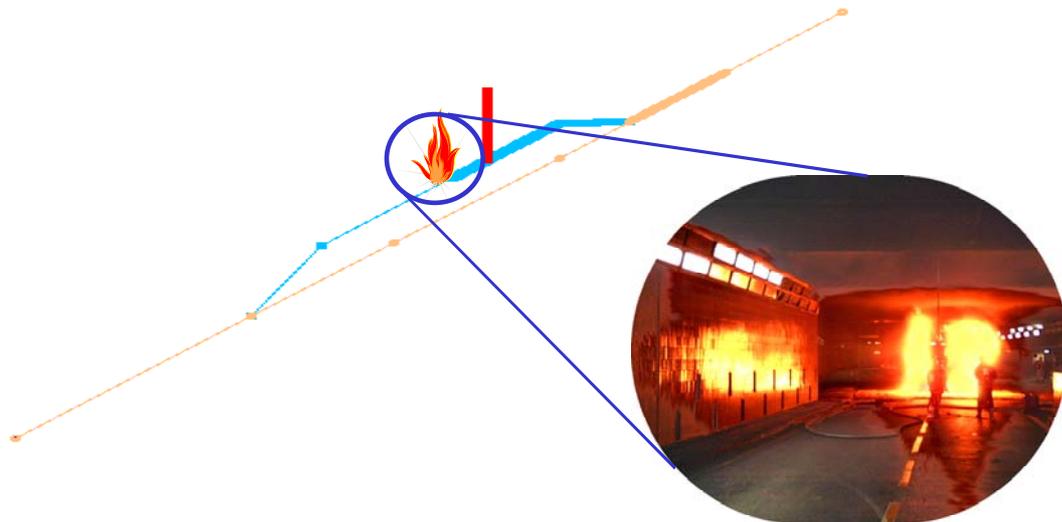
Figure 1—1: Illustrated example of the 'overall phenomena' concept affecting a ventilation network and the 'local phenomena' relating to an accidental occurrence.

In the first approach, it is possible to analyze the simulation of the effects of accidental gaseous releases in underground networks in the same way: the need to simulate different overall and local physical phenomena which therefore call on different simulation tools.

A number of different kinds of tools partially meet requirements when it comes to modeling the previously mentioned effects of a fire [1], [2].

In theory, CFD (Computational Fluid Dynamics, [3], [4]) models allow describing all phenomena as they can include combustion modes and heat transfers by fluids and solids. They are however only usable in simplified cases (run-off areas of ranging from a few tens to few hundreds of meters) for calculation time reasons and because of the time it takes to prepare these calculations. Furthermore, these simulation tools require extremely careful use as well as high levels of expertise.

To overcome the calculation time problem, simplified models, initially developed mainly in a building fire context, may be used [5]. These so-called 'zone models', offer a simplified description of smoke stratification phenomena and may take into account areas of heat release, jets, stratified layers… They are based on an empirical analytical description of the different zones within the simulated calculation area, hence their name. The main complaint against them is that they do not take into account all of the physical phenomena of the fluid emissions from the event, generally, they do not take into account the amount of motion results or the non-stationary behavior that applies and they are therefore reserved for the special cases for which they have been validated. In these cases, they may represent an interesting alternative approach to the CFD models, depending on the level of precision required. They generally require calculation times that are far less than those required by CFD models, so that they can be used to approach the simulation of emissions from large size confined spaces.

However, neither the CFD models, nor the zone models are able to simulate the balancing of a complex aeraulic networks, possibly comprising a description of many hundreds of tunnels. The networked single dimensional models (1D models) are made for this. They are initially developed to handle mine shaft ventilation problems (this was the case at INERIS, [6]), mains networks (water, gas, etc.) or for managing hydrographic networks. They are based on a single dimensional simplification of general Navier-Stokes equations. They make it possible to make a simplified allowance for the effects of motors or power dissipaters for network fluid flows. They also allow the integration of additional terms into the equations so as to make allowance for specific factors that may be present in the ventilation networks, such as fans, air locks and, in a simplified way, for accidental effects.

Consequently, to compensate for the problems involved in correctly and simultaneously correcting for local and overall effects - as mentioned previously - it is therefore natural to think about how to put to use the advantages of each type of calculation codes as described briefly above: CFD models, area models and 1D network models.

Any convergence of these advantages passes through what we will later refer to as **coupling**. The term coupling refers to a link, one that is technically a computer link between two different calculation codes and that in phenomenological terms allows a suitable physical simulation of the various areas in an overall flow.

The coupling concept represents a fundamental point in this project, around which the main work undertaken is built.

Another characteristic point of the accidental effects, is that they are transitory by nature [7]. This is because an accidental phase during the operation of a structure passes through:

- a preexisting context,
- the occurrence of an accident and its sequence,
- the protective actions implemented once the accident is detected,
- control over the phenomena triggered by the accident and allowing a return to a normal situation.

All of these phases represent an accident **scenario**. Correctly defining the operational actions to be implemented during a scenario is a second fundamental point in restricting the effects of any accident that takes place in an underground structure.

Consequently, and right from the start, the project integrated this goal which requires the ability to take into account highly diverse physical effects likely to change the ventilation in an underground structure. In practice, depending on the type of construction involved, it is necessary to be able to allow for actions affecting the mechanical ventilation system and various other mechanical components as well as the movements of objects that may create air movements (elevators, trains, automotive traffic… in addition to natural effects such as thermal streams or even differences in atmospheric pressure from one place to another within the structure.

The events that occurred during this project (accidents in the Mont Blanc tunnel, in the Tauern tunnel, the Kaprun track railway) provided a dramatic illustration of why it is so important to correctly manage an accident scenario and therefore of the tools that can contribute to predicting the thermal or chemical effects that may occur during such events.

These accidents served as triggers for a rapid change in the French regulatory context. Consequently for road tunnels, the following changes were implemented:

- the release of interministerial circular No. 2000-63 dated August 25, 2000 relating to safety in tunnels (of more than 300 meters in length) which calls for new procedures for authorizing, opening and monitoring French road tunnels, and especially the need to carry out a Specific Danger Study (or ESD from its French initials),
- the release of interministerial circular No. 2000-82 dated November 30, 2000 relating to regulations that apply to movements of vehicles carrying dangerous merchandize (VTMD) in French road tunnels, and which especially calls for undertaking a Comparative Risk Analysis (or ACR from its French initials) regarding carrying dangerous materials over any given route,

The direct consequences of the application of these circulars are, in addition to the need to produce an ESD and/or ACR, the:

- setting up of a Road Tunnel Safety Evaluation Committee (CESTR), the centerpiece of the new regulatory approach, the body charged with issuing a ruling on safety management in these tunnels,
- allowance for, as part of the structure's safety file, of an expert assessment on the Specific Danger Study as part of the safety file on the structure.

The implications of these new regulatory measures in the transportation field are numerous and require changes to the resources applied and the assessments made in terms of Specific Danger Studies. This project appears therefore, in this context, as a research effort in order to develop new tools that are able to help with producing a Specific Danger Study, and especially in evaluating the effects of fire and gaseous emissions type accidents.

Note also that proposed legislation covering the safety of transportation infrastructures and systems is being prepared (METL - press release dated March 21, 2001) and this could extend the current requirements for road tunnels to rail tunnels too.

In the more general context of underground structure safety, and just as importantly, this project proposes developments that may be of use in studying reversible or irreversible storage sites for radioactive waste in deep geological storage locations (law dated December 30, 1991) or also the prevention of minor hazards during operation of after ending operation (legislated dated March 30, 1999).

Furthermore, it also offers the option of evaluating the behavior of certain pipe networks present in industrial environments, such as for example, an evaluation of leaks from gas carrying networks in confined locations or even a performance evaluations for fire hydrant supply mains.

Based on these observations of the project's fundamental goals (coupling) and the potential context of use of any such future tool, the project was arranged to include the following work:

- the creation of a new (1D) code for ventilation network calculations – New-Vendis[1] – with the right architecture for exchanging information with an external calculation code (whether a three-dimensional CFD or other approach) and allowing numerous accidental and natural effects to be taken into account, and which is adaptable to different usage contexts,

- the implementation and perfecting of the 1D/3D coupling and therefore the definition of the exchanges of information that need to take place between the two calculation codes in order to achieve an overall simulation of the network that suffered the accident. This work first requires defining the computer techniques that allow the exchanges of information detailed here.

- the physical validation of the results provided by the 1D tool alone and the 1D/3D coupling comprising a number of important points:

  - numerical and experimental comparisons of the results provided by the 1D tool, i.e. the network balancing calculations,

  - creating a new small scale experimental means,

  - performing experiments dedicated to validating 1D/3D coupling,

  - performing numerical and experimental comparisons relating to 1D/3D coupling.

Readers will find that this report provides a presentation of the different kinds of work undertaken to achieve these goals, set out in line with the four main sections listed below:

- reminder on the goals and choices inherent to this project,

- summary presentation of numerical work,

- summary presentation of experimental developments,

- summary presentation of digital and experimental validation work.

Lastly, it is worth underlining that this project, which approaches new concepts as regards the simulation of the accidental effects of fire and gaseous emissions in confined locations, has a research aim. Consequently any developments made here and the results obtained cannot be directly applied in an engineering context. Nevertheless, the developments made as part of this project were conducted in view of achieving short term progress directed towards the needs of applications.

---

[1] Also called a '1D tool' or '1D model' or 'simulation tool' in the remainder of this document.

2. REMINDER ON THE GOALS AND CHOICES INHERENT TO THIS PROJECT

2.1   REMINDERS AND CHANGES TO GOALS

Very summarily, this project mainly comprises:

1. perfecting a tool for simulating accident situations in underground networks called: New-Vendis,

2. achieving, and perfecting, the (1D/3D) coupling between the two calculation codes (New-Vendis and Phoenics),

3. wherever possible, validating these developments through numerical, theoretical or experimental means.

Like all projects, this project had to face a number of unforeseen events, whether of an internal or external nature. The main ones being, in addition to any technical difficulties relating to the setting up of a new reduced scale test laboratory, the significant participation by INERIS in post-accident management and the benefits, from a regulatory point, of the Mont Blanc catastrophe (taking part in the technical and administrative investigation, taking in part in undertaking fire tests during the criminal investigation, taking part in writing circular number 2000-63 dated August 25, 2000 calling for the undertaking of Specific Danger Studies…). All of these different unforeseen events caused the scheduled program to be carried out over a effective period of some two years instead of three. This reduction in the periods available for carrying out practical work led to a revision of the goals stated above, with the following impact:

- regarding work on coupling:

    - the principles provided for achieving coupling were studied,

    - the programming and validating of data exchange modes between the calculation codes were developed and tested,

    - the physical validation of the 1D/3D coupling was not covered;

- regarding experimental tests:

    - the means used to produce validation data are operational and were qualified,

    - some of the tests required for validating a 1D/3D computer coupling were undertaken, but additional testing will be required,

    - tests were mainly used with the aim of perfecting the experimental process and to ensure that usable data was obtained for validating the code coupling.

2.2 CHOICES RELATING TO NUMERICAL DEVELOPMENTS

Part of the work comprises being able to precisely calculate the ventilation in a structure using a **single-dimensional model** of a tunnel network. This 1D modeling must also allow the detailed definition of an **accident scenario**. First of all, it should be restated that the accidental effects taken into account exclude explosions. The interest shown therefore relates primarily to dangers such as fires or massive releases of gas. For both of these effects, the aim is therefore to represent, as finely as possible, the propagation of fire smoke or gas within the underground network. The modeling of the underground network takes into account all of the physical effects likely to influence the behavior of this network. These influencing parameters vary over time so as to gain data on network dynamics and on accidentals effects which are by essence, transitory.

Another goal that needs to be achieved by the simulation tool is to have a part of the tools able to **finely represent the accident zone**, a goal that can be achieved by using **three-dimensional code** (3D) which provides a better geometrical and physical description of the local phenomenon observed and as a result, its impact on the balance of the entire ventilation network. To develop this simulation, the choice was made from the start of the project to take an interest in the description of the three-dimensional accident zone using CFD (Computational Fluid Dynamics) type calculation code. These calculation codes allow resolving equations that cover fluid motion: Navier-Stokes equations. As stated in the introduction from the example of fire, this kind of code is not the only one that can be used to refine the behavior of the area directly affected by the source of the accident (zone models can also be envisaged), but this kind of code is widely present in the market and already validated. Furthermore, some of these codes offer the ability to add user programming which is taken into account in the basic commercially distributed product. Both of these aspects are important for the entire project, and especially the development of data exchanges between the single-dimensional part of the simulation tool and its three-dimensional part. Consequently the **Phoenics** calculation code was chosen to approach the simulation of three-dimensional effects.
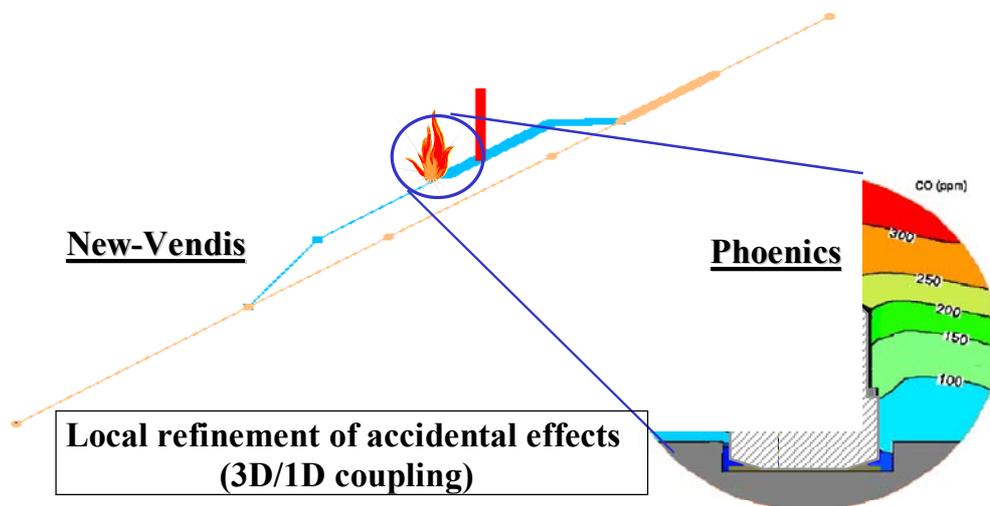
Figure 2—1: Local three-dimensional calculations performed from Phoenics.

We are seeing fast and continuous progress in computer processing power. Simply stated, since the start of the project (in 1997), the internal clock speed of office computers has been multiplied by at least a factor of four. This fast changing level of computer power has influenced digital developments in that it becomes envisagable to bring CFD codes into the evaluation of accidental effects on underground networks while still using another application to calculate the overall behavior of the ventilation systems for these networks. This approach will be reinforced in future as computer power will continue to increase over the coming years.

Consequently, the evolution linked to Intranet and Internet networks facilitates digital developments where there are communication requirements. Regarding this point, the project has an obvious need: handling data communications between a 1D calculation code and an external (in our case, 3D) calculation code. It was therefore useful to allow for a development platform that allowed the use of these Internet connections. Another alternative was to develop both of the calculation codes (1D and 3D) to better manage their data exchanges. This option was quickly dropped at the start of the project, as this approach would have required large scale software development work whereas it should be possible to achieve 1D and 3D code coupling using off the shelf 3D code.

Consequently, an easy to use development platform was sought, allowing the management of a major computer based project which leading to the quick and easy ability to use these developments. For these reasons, and because INERIS had already gained significant experience of major computer based development work using this platform in the field of data acquisition for atmospheric dispersion, an application called LabView was chosen. This application allows developing a program with an easy to use (graphic) user interface while offering functions such as: math libraries, virtually transparent portability from a Windows system to a UNIX system, the ability to manage computer networks and client/server applications and to manage data acquisition systems…

Lastly the simulation tool should be able to allow for the data inputs from both the calculation of accident scenarios affecting the ventilation network (single-dimensional) and fine simulation of the accident area (three-dimensional fire smoke and gaseous emissions). This ability to take into account both 1D and 3D simulation in the overall simulation is what we refer to as coupling the 1D and 3D simulations. This coupling function has been built-into the New-Vendis tool itself. New-Vendis therefore integrates both the balancing calculations for a ventilation network as well as the ability to accept code coupling based on its orchestration of the necessary data exchanges.
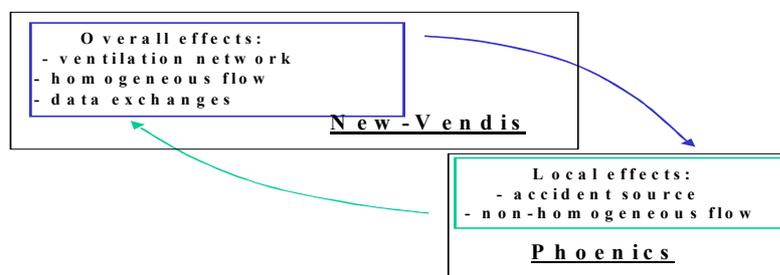
Figure 2—2: Diagram presentation of the respective roles of New-Vendis and Phoenics.

All of the digital development work is presented in greater detail in section 4.

2.3   CHOICES RELATING TO EXPERIMENTAL DEVELOPMENTS

As introduced previously, the experimental developments have an important goal in validating physical phenomena that can be simulated by coupling different types of calculation codes. This is of course an extremely fine level of validation where it is necessary to be able to estimate errors from the simulation of a ventilation network, those that come from the simulation of the three-dimensional calculation area and those from the representativity of the coupling effects, without forgetting the uncertainties inherent to the experimental process itself.

Objectively, this level of validation requires perfect management of validation data, and hence experiments. The choice was therefore made to develop, at INERIS, a small scale model allowing this kind of validation and an experiment dedicated to validating 1D/3D coupling.

These experimental developments are described in section 5.

More generally, INERIS choose to develop the small scale simulation activity as this activity is a good way to complement the exploration of full scale test applications as well as a good way to obtain digital validation data. The experiment dedicated to validating the digital development data for this project was therefore installed in a laboratory specially designed and built during this same project.

The complete installation is also presented in section 5.

2.4   CHOICES RELATING DIGITAL AND EXPERIMENTAL VALIDATIONS

As the problem was to fully develop a new calculation code, the validation work was undertaken for the various basic developments along two lines:

- Physical model validations from experimental results gains from small or large scale experiments,
- Digital resolution method validations from theoretical or bibliographical analytical results.

As a general rule and based on the layout set up for the simulation tool, validations relating to goal 1 (refer to 3.1) mainly covered validating the implementation of the existing physical models and their integration into the overall tools rather than on the development of new physical models.

Regarding validations relating to goal 2, in the end, these primarily comprised implementing a new experiment specifically developed for this goal (refer to section 5). The validation work for this goal also covered the data exchange methods applied to perform the digital coupling between New-Vendis and Phoenics. The digital code coupling is presented in sub-section 3.5.3.

A part of the validation work is presented in section 6.

3. SUMMARY PRESENTATION OF NUMERICAL WORK: NEW-VENDIS

Originally, starting in 1976, INERIS developed a program for calculating ventilation in underground networks, called Vendis [6]. This program, in its latest version, is able to simply simulate a fire. However, and as was stressed in the preceding section, computer systems have evolved greatly and goals set out for this project are not compatible with a simple revamping of the *Vendis* code. Consequently, the decision was taken to create New-Vendis by expanding the capacities of the former calculation code and integrating the goals of this project into it as best possible.

The main characteristics of the New-Vendis tool created during this project are therefore described hereinafter.

3.1 NEW-VENDIS NUMERICAL STRUCTURE

The structure of New-Vendis is designed to be able to take into account the simulation of overall effects (ventilation networks) along with local effects (accidental effects). From the start of the project, the goal of code coupling was included in its structure. The diagram below shows the structure of Vendis (the former tool) and that of New-Vendis.
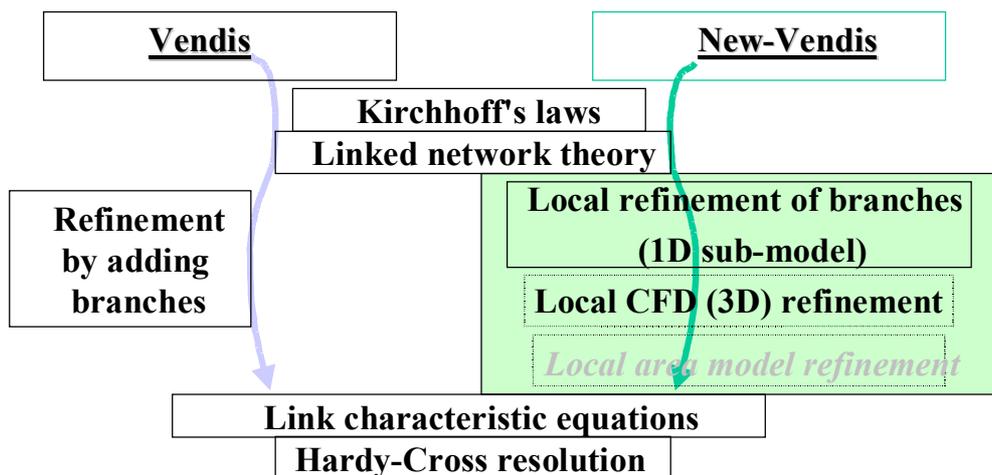
As we can see in both tools, any local refinement of the description of the effects of an accident source affect a central calculation code part. In the *Vendis* case, the solution is refined by adding contiguous branches to the linked network. In this way the number of branches to be taken into account when resolving the calculation may become significant. Consequently, in *Vendis*, it is complex to take into account any data from another code as refining the solution does not call on any model other than the general 'propagation' model that covers the flows between network branches. It is not therefore possible to call on a specific physical model for one branch and on another model in another branch.

The latter option is built-into New-Vendis. Consequently, the tool's structure is more complex but it allows the solution to be refined locally using different models. In fact, in New-Vendis the network is primarily built up from the branches that are absolutely necessary for describing the network and the nodal points, associating the use of local refinement to the evaluation of the solution in the desired branches.

This kind of local refinement is obtained by turning the branches into discrete data (using a 1D refinement model) or an external model. In both case, and this is a concept that is far different from *Vendis*. Refining the solution at one part of the network is consequently the same as transferring the data from the local refinement effect to the affected branch nodes. As a result, conceptually, the data produced by a local solution refinement model is transferred to the resolution method used for conventional network grid systems.

Two kinds of refinement models were considered during this project:

- local refinement using a 1D model, which will be called 1D/1D coupling,
- local refinement using a 3D model (1D/3D coupling).

Structurally, integrating the concept comprising bringing data back to the branch nodes should make it possible to use another local refinement model, such as zone models for example.

3.2 NEW-VENDIS MODULE FUNCTIONS

From the user's point of view, the general structure of the calculation code used is no different than most simulation tools and comprises a:

- pre-processor,
- processor,
- post-processor.

Initially, users describe the ventilation network by declaring motor or energy dissipation effects for drafts and smoke. Then, local calculation refinement is required, either through the 1D/3D coupling or by converting the zone around the accident source into discrete data while still staying within a single-dimensional exhaust air flow description (1D/1D coupling). This description of the calculation case is performed using a pre-processor (integrating a grid mechanism) that is described in sub-section 4.2.1.

Then, in a second phase, a scenario is simulated. The user will then calculate the scenario described using the pre-processor with the help of a processor that allows defining overall calculation parameters such as for example the reference temperature or pressure levels, the simulation time or the definition of result storage. This processor is described in sub-section 4.2.2.

Lastly, users can call on a series of small utilities that allow various analyses of the calculations performed. Together these small utilities will be referred to as the post-processor in the remainder of this document and they are described in sub-section 4.2.3.

3.2.1 Pre-processor

3.2.1.1 General

The pre-processor performs two functions: describing the geometry of the branch network that makes up the complete underground network and declaring the motor or dissipating effects that affect air flow currents.

Let us start with a description of geometry. First of all, the aeraulic network is taken to be a set of branches inter-connected by nodes. Each branch is therefore defined by a departure point and an arrival point, with internal points used to refine the geometrical description of the specific branch. The parameters applied for each branch also comprise:

- its name,
- its hydraulic diameter,
- the perimeter of its straight line section,
- factors used to calculate momentum losses and heat transfers.

An example of a dialog box used to declare a network branch is provided below.
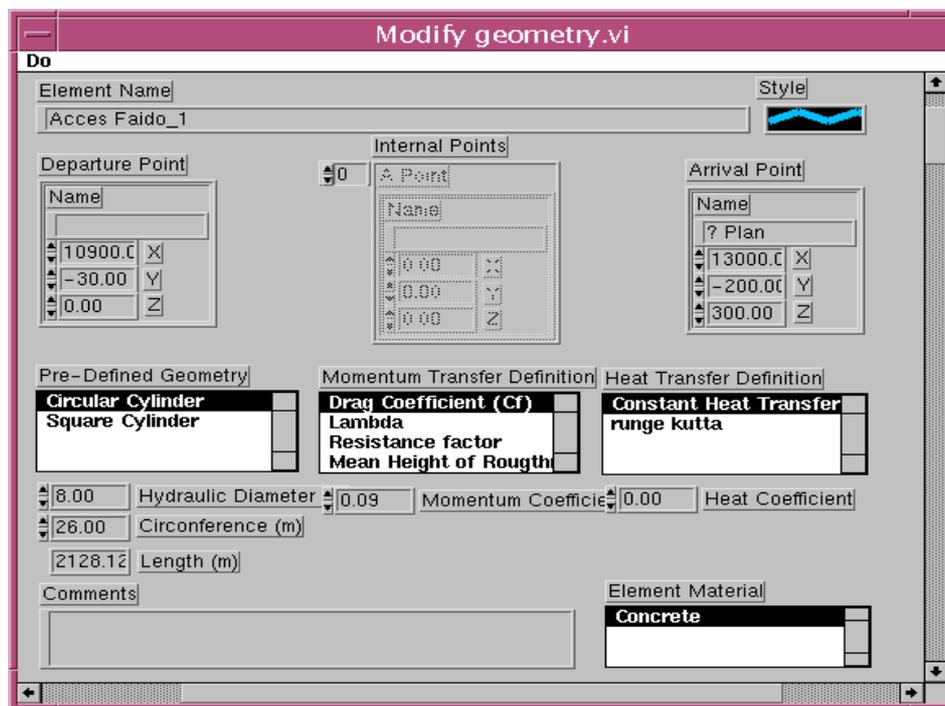
Figure 3—1: Interface for defining the branches that make up the network.

Other faster methods for declaring network branches are built-into the grid definition mechanism such as the use of grid definition mechanism such as the use of cursors and methods for duplicating identical branches, adding/deleting branches or discrete conversion points for a branch, etc…

The number of network branches is only limited by the amount of memory available in the computer. It should also be noted that this is true for all of the functions provided by the calculation tool. This is because the software chosen for development purposes (LabView), handles dynamic memory allocation tasks and does not require preset data table sizes, unlike the programming language (Fortran 77) used formerly.

Here are two examples of ventilation networks: a twin-tube rail tunnel and a mine shaft network.
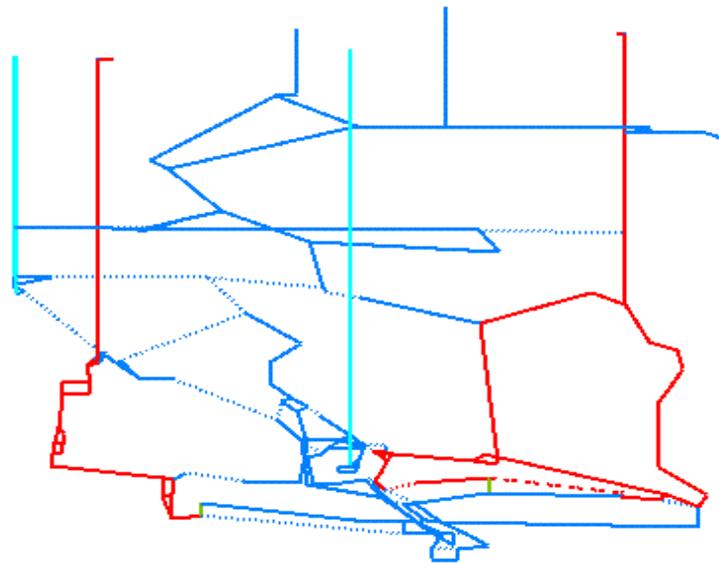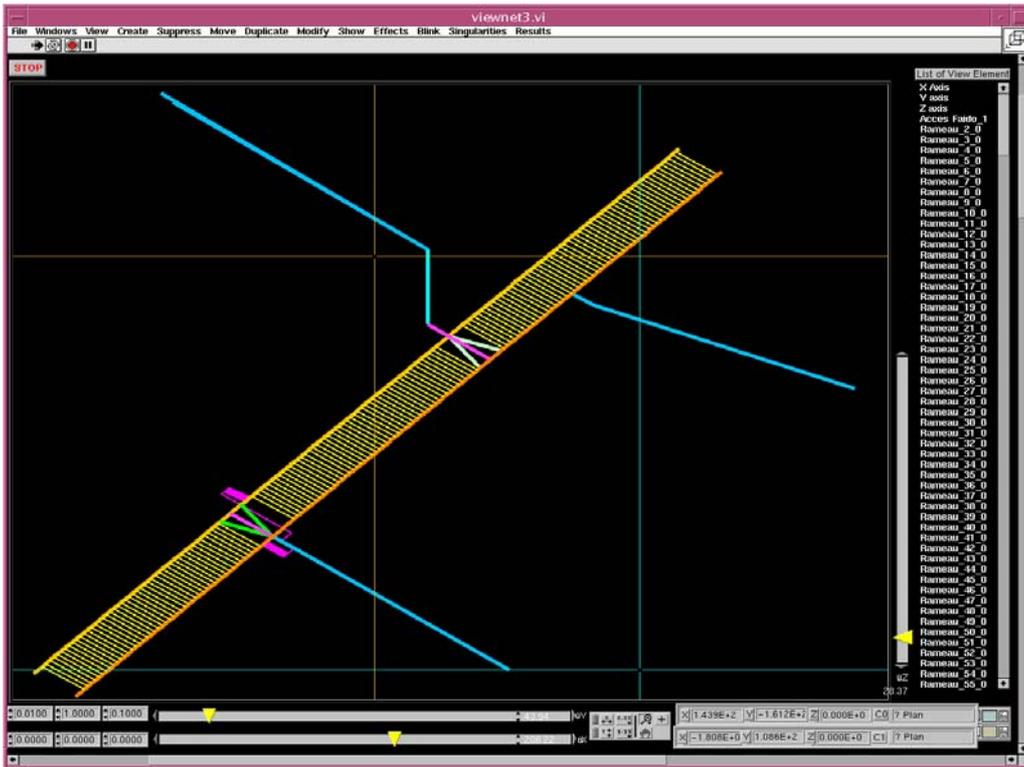
Figure 3—2: A 'twin-tube rail tunnel' and 'mine shaft ventilation' aeraulic network case.

Consequently, from the pre-processor, users can perform all of the operations required for describing ventilation network tunnels, even complex ones.

Users can also describe the motor or dissipater effects on the air outflows through the branch network. These effects are filled-in using two pointers to the affected branches. The presence of two pointers makes it possible to describe 'oriented' or 'non-oriented' effects as well as 'mobile' or 'dynamic*' effects during simulation. The term "mobile effect" applies to effects that act on different network branches during simulation, for example the movement of a solid that creates a piston effect in the branch where it is located at a given time and then creates a similar effect in an adjacent branch a moment later.

Effects with a positioning that does not vary over time are easy to declare. The mobile effects are a little more complex to define for it becomes necessary to define within the ventilation network, those paths that the mobile effect may cover. The pre-processor offers the ability to define these paths. For now, we have only looked at the description of piston effects caused by a moving object moving along the ventilation network, but the basis for forthcoming descriptions of other mobile effects have already been laid out.

Effects are therefore defined with the help of the pre-processor, in the tunnel network that the user defines first. In order to widen as far as possible, the scope of possible investigations using this project and its potential future applications, we took an interest in identifying fluid networks that it would be possible to study and their main characteristics in terms of motor or dissipater effects on fluid movements through these networks.

Among the different types of structures, we retained:

- mines,
- underground storage facilities,
- road tunnels,
- rail tunnels,
- industrial processes at atmospheric pressure (dryers, furnaces…),
- low pressure pipe networks (gasses, liquids).

For these different types of structures, we allowed for the main natural, environmental or operating effects to be taken into account:

- wall drag friction, heat transfers and gaseous mixes,
- equipment and service action effects (fans, pumps, air locks, baffles…),
- moving equipment effects (e.g. elevators), traffic effects (road or rail),
- atmospheric effects that induce natural air currents, like temperature or pressure differences between different points in the network and outside.

Regarding accidental effects, we considered the possibility of simulating:

- fires and the gaseous releases developing in the network,
- gas or liquid leaks from a network.

Then the calculation code structure was thought out so as to be able to homogeneously declare the different effects (whether natural, environment or operations related, or accidental) to be taken into account in network as well as their modeling in the calculation.

---

* Dynamic effects are those that users cannot completely define for they are dependent on the calculation history and the solution found during the previous time increment.

To this end, the various effects are defined as text files that include the description of the variation in time of each effect considered during the simulation process (an effect scenario). In the aeraulic network dynamics equations, these effects are then translated in the form of additional terms to add to the ones present in the basis equation (single-dimensional equation for the movement quantity) with the help of physical models that influence one or more airflow variables:

- velocity,

- pressure,

- fluid density, possibly dependent on:

  - temperature,

  - chemical make up of the fluid circulating in the aeraulic network.

Originally, every effect integrates a **scenario** concept. A scenario translates the non-stationary nature of the effect to be taken into account, such as for example the starting of a fan, the opening of an extractor hatch…, phenomena which are important when it comes to modeling an accident situation which is by definition **transitory**. Some effects, the so-called mobile effects, are additionally variable in space, i.e. their position within the ventilation network varies over time.

The typical format type of these effect scenario files is shown below:

Information line on the effect				
Effect type	0	Effect sub-type	0	
Information lines on the effect				
	"			
Information lines on the effect				
	Parameter 1 variable in	Parameter 2 variable in	Parameter 3 variable in	
time (s)	time	time	time	etc…
0	100.00	1.00	2.50	
60	100.00	2.00	2.50	
90	100.00	3.00	2.50	
120	10.00	4.00	2.50	
10000	50.00	5.00	2.50	

As we can see, the effect files comprise:

- information on the type and sub-type, where the sub-type generally represents a variant of the effect type,
- information lines that can, for example, comprise numerical parameters that state values (that are constant during the simulation) that are useful in characterizing the effect on the physical model in question, and
- lastly, columns that provide values that are useful in characterizing variations of the effect over time during the simulation (effect scenario).

A general principle has been adopted. Every effect is cumulated in the equations that characterize the branches like a source term added to the basic equation, with the exception of so-called 'imposed' effects.

The imposed effects are used to set in for a given branch, any arbitrarily desired characteristics (e.g. velocity, concentrations or temperature), these variables are not then the result of any search for the solution to a characteristic equation or the ventilation network balance, but are directly 'forced' to the desired value. The ability to arbitrarily set the main characteristics of the air flow out of the branches can be used in setting up the 1D/3D coupling (refer to sub-section 3.5.3.).

The other effects (in the general case) each contribute to an air flow solution through the resolution of a system of independent equations. This report does not go into a summary regarding the numerical resolution of the equations [8], [9], [10].

3.2.1.2 Natural and Environmental Effects

The various natural, environmental and operating effects developed in New-Vendis are covered below.

3.2.1.2.1 Imposed Pressure Jump in a Branch

Note that the positive direction in a pressure jump in a network branch (an increase in pressure) is defined by the user in the pre-processor using the cursors. Users can, as part of a scenario, impose pressure jumps that comprise both positive and negative values. Consequently natural or mechanical (using equipment that generates a pressure jump) thrust can be simulated using this effect (a valley effect), inducing a draft through the branch. This effect is added to the other motor effects present in the affected branch and into the network, playing a part in its general balance.

Pressure jump			
Effect type	3	Effect sub-type	0

Time (s)	DP (Pa)
0	7.5
200	40.0
500	70.0
10000	7.5

3.2.1.2.2 Fan

The way fan effects are taken into account is declared in the pre-processor using the following file:

```
Fan
Effect type                      0        Effect sub-type              0

Maximum speed (rpm)           1500  Minimum speed (rpm)          400
Fan characteristic at         1500  rpm
Reference density             1.17  kg/m3
Q (m3/s)                         0                  100    150    175    200
H (mmCE)                      1.34                  1.34   1.34   1.34   1.34
Time (s)                   rpm
                       0     10
                      50    200
                     100    600
                     150    800
                     200   1000
                     300   1200
                   10000   1200
```

Note that the 'fan' effect allows recording the main characteristics of this type of machine:

- taking into account a velocity/pressure curve that is characteristic of the fan at a given speed and with a given reference density.
- Retaining the flow volume that passes through the fan at a given speed, regardless of the density of the fluid passing through it.

Just like for other effects, the scenario is described by the variation in the ventilation rate for which the corresponding characteristic curve is calculated, on the basis of the characteristic curve provided by the user for the reference rotation speed.

3.2.1.2.3 Singularity Function

This effect is used to allow for a load loss function in a network branch, depending on the dynamic pressure value and the direction of the airflow in the branch. This effect is used to simulate the load loss characteristics of certain singularities such as leaks, sudden reductions/increases in width.

There are a number of singularity characteristics. In this project, no effort was made to implement all of the characteristics of any given mechanical component, but rather to provide the ability to define a general polynomial that could be representative of the required singularity. Consequently the singular load loss is modeled using the following function:

$$\Delta P = \zeta \ P_{dyn} = \sum_i a_i x^{b_i} \ P_{dyn}$$

$$\text{with } P_{dyn} = \frac{1}{2} \rho u^2$$

ζ = singularity (no dimension),

a_i, b_i = constants and exponents in the singularity function,

x = parameter that varies over time, representing transitory effects.

Singular load loss in a direction and dependent on Pdyn			
Effect type	3	Effect sub-type	1
zeta+ (a_i+, b_i+, ...)		0.5	0
zeta- (a_i-, b_i-, ...)		1	0
Time (s)	x		
0	1		
60	1		
61	1		
62	1		
2401	1		
3000	1		

This function is used to define complex or relatively simple singularities, like the example presented here. This example represents the effect of a sudden widening in one direction and a sudden narrowing in the other direction. This is a simplified way to allow for the tunnel head effect in a simulation where the direction of the airflow through the tunnel is most likely unknown.

We should however, note the need for care when using this effect for a significant singularity may lead to suddenly upsetting the flow through a network (opening/closing a door) and make the calculation solution an unstable one. Regarding this point, numerical methods would allow resolving these problems but this work was not covered in this project.

3.2.1.2.4 Mobile Piston Effect on a Path

Setting up a mobile piston effect calls on the path concept. The pre-processor can be used to define a path as a succession of zones[2] in a given direction. The effect is therefore considered to be mobile along a path. The effect scenario defines its motion speed in time and therefore indirectly its position along the path. We know to which branch of the network and to which part of the network, the motion effect applies.

This effect is expressed by using a basic effect such as a pressure jump pour the piston effect.

The mobile piston model was developed based on the one proposed for trains by Maeda *and al* [11], with:

$$D_{tr} = \tfrac{1}{2}\rho S_{tr}\left\{(C_{tr} + R)\left(\frac{u - u_{tr}}{1 - R}\right)^2 - \frac{\lambda l_{tr}}{d}\frac{(u - Ru_{tr})\cdot|u - u_{tr}|}{(1 - R)^3} + \frac{\lambda_{tr} l_{tr}}{d_{tr}}\frac{(u - u_{tr})^2}{(1 - R)^3}\right\}$$

with:

- $R = S_{tr}/S$ the effective cross-section ratios of train and tunnel,

- ρ the air density,

- C_{tr} the train's drag pattern factor,

- u and u_{tr} respectively: air speed and train speed,

- λ, λ_{tr} tunnel and train friction factors,

---

[2] In this context, a zone is a fraction of a path. This concept is therefore very different from the 1D/3D coupling area context.

- d, d_{tr} tunnel and train hydraulic diameters,
- l_{tr} train length.

The previous equation is used to calculate D_{tr} the train's aerodynamic drag. In [11], u is obtained using an ancillary set of formulas. In New-Vendis, the velocity u of the air in the tunnel is evaluated directly by New-Vendis.

This model will not give realistic results when the air flow velocity away is similar to train speed. This is because for the air to move away almost as fast as the train, the calculated pressure difference becomes strongly negative. This is not physics for the train still has a drawing effect on the air. Although the air is accelerated by the reduction in cross-section in the ring section between the train and the tunnel, the train remains the driving force over a wide speed range. The model in Maeda *and al* was developed to represent high speed train traffic. The limit case where the train speed is of the same order as the air speed is therefore outside of the range of validity of this model.

We therefore connected this model to a simpler model for low differential speeds. To estimate the pressure variation induced by the mobile body, when slight speed variations take place, a simple load loss calculation is performed. The cross-section considered is the open cross-section between the mobile body and the tunnel wall.

$$D_{tr} = \tfrac{1}{2}\, \rho\, R\left(u - Ru_{tr}\right) \cdot \left|u - u_{tr}\right| \left(\frac{\lambda_{tr} l_{tr}}{d_{tr}} + \kappa_{front} + \kappa_{rear} \right)$$

with:

- κ_{front} and κ_{rear} the singularities of the front and the rear of the train.

The two equations set down this way are connected at their 'crossover point'.

Below is an example of a declaration file for a scenario with a train moving along a path that travels the ventilation network declared by the user (path 1 in the example).

Train motion scenario						
Type		6	Subtype		0	
Vehicle length (m)	Vehicle cross-section (m²)	Hydraulic diameter (m)	Vehicle resistance (Cf)		Front factor	Rear factor
300	6	2.5	0.005		0.5	0.5
Relative Time	Way	Velocity m/s				
0	1	0				
100	1	10				
500	1	20				
1000	1	20				
1500	1	10				
2000	1	0				

3.2.1.2.5 Model for Road Traffic in Part of the Network

Large number of vehicles traveling through a road tunnel is a more diffuse and 'distributed' phenomena that a train passage, and more generally that of any single solid object. This is because the large number of vehicles and distribution throughout the tunnel is not a local phenomena, despite it remaining located within those branches that allow traffic.

The positioning of this traffic in the network is performed in the same way as when setting the piston effect of a mobile object, i.e. by using the path concept (the path followed by road traffic). The effect is not by nature a mobile one as it does not move within the network. However, the value of the piston action may vary in time. Creating any traffic congestion clearly illustrates this problem. The effect must therefore be reevaluated at every time increment, based on the results from the previous time increment. This is then called a dynamic effect.

Traffic is represented on an average basis. To do this, vehicles are sorted by class. The format developed here only differentiates between two classes, light passenger vehicles and heavy goods vehicles. It remains possible to extend this approach to a greater number of classes while still retaining the same calculation diagram.

The density of vehicle presence allows describing the traffic for each class. This will allow obtaining the effect produced by all of the traffic based on the effect of just one vehicle. The pressure variation caused by a single vehicle is defined as:

$$\Delta P = \tfrac{1}{2}\,\rho\,\frac{S_v}{S_t}\,Cx\left(u_v - u\right)\left|u_v - u\right|$$

with:

- ΔP pressure variation caused by vehicle motion,
- S_v and S_t vehicle and tunnel cross-sections,
- Cx vehicle aerodynamic drag factor,
- u_v and u vehicle speed and air speed within the tunnel,
- ρ tunnel air mass.

The tunnel cross-section is generally large in relation to the vehicle cross-section, which makes it possible to ignore any interaction with the tunnel walls. This approximation is satisfactory for cross-sections such as: $S_v\big/S_t \leq 0.2$ if one further considers that the space between vehicles is sufficiently great so that any effect caused by a vehicle does not modify the effect of the following vehicle.

The piston effect caused by vehicle traffic is determined by summing the contribution made by every vehicle present in the affected part of the tunnel.

Below is an example of the declaration file used for a road traffic scenario.

Vehicle traffic scenario								
Type	6	Sub-type	3					
LV c-section (m²)	LV Cx	LV length (m)	HV c-section (m²)	HV Cx	HV length (m)			
2	0.4	8	6	0.8	20			
Relative Time (s)	Veh. density+ (veh./km)	LV speed+ (m/s)	HV speed+ (m/s)	HV ratio+	Veh. Density- (veh./km)	HV speed- (m/s)	HV speed (m/s)	HV ratio-
0.00	1.00	19.44	13.89	0.16	1.00	19.44	13.89	0.16
10000.00	1.00	19.44	13.89	0.16	1.00	19.44	13.89	0.16

3.2.1.3 Accident Effects

3.2.1.3.1 Imposed Velocity in a Branch

Imposing a given flow in a branch requires specific processing at the numerical level that will not be developed here.

Below, an example of a file is provided for taking into account, within a network branch, a flow rate with an intensity that varies over time. The positive flow direction is set using pre-processor cursors when the user requests that this effect be taken into account in the branch.

Imposed flow effect			
Effect type	4	Effect sub-type	0
Time (s)	Flow (kg/s)		
0	-50		
60	-20		
61	50		
2400	100		
2401	200		
3000	200		

3.2.1.3.1.1 Imposed Flow and Concentration within a Branch

This effect is used to perform a simplified simulation of a massive gas leak or a mixture of gasses. This effect therefore takes into account a mass flow rate for the gaseous release as well as the make up of the gas mixture. To do this, the properties of different pure gasses must be filled-in in New-Vendis. To date, there is not as yet a true database for gaseous products in New-Vendis. However the laws used for calculating the properties of gaseous mixtures have been implemented so as to be able to simulate the injection of gaseous products into a network, and to be able to, at any point in the ventilation network, recalculate the basic physical properties of the gaseous mixture, i.e. composition in terms of mass concentrations, specific heat levels, viscosity, density….

A gaseous release is simulated in two stages:

- first of all, the user creates a fictitious injection branch,
- then, and for this branch, the user declares an 'imposed flow and concentration' type effect.

The calculation tool will then taken into account the desired injection by reading the injection scenario that the user prepared ahead of time.

The effect scenario in this case covers the flow and make up of the gaseous mixture which may vary over time.

Imposed flow and concentration													
Effect type	4		Effect sub-type	1									
Scenario						Concentrations							
Time (s)	Flow (kg/s)	Dryair	O	O2	Nitro	Helium	CO2	CO	H2O	Liq	SO2	HCN	HCL
0	20		0	0	0.3	0.7	0	0	0	0	0	0	0
60	20		0	0	0.3	0.7	0	0	0	0	0	0	0
61	20		0	0	0.3	0.7	0	0	0	0	0	0	0
2400	20		0	0	0.3	0.7	0	0	0	0	0	0	0
2401	20		0	0	0.3	0.7	0	0	0	0	0	0	0
3000	20		0	0	0.3	0.7	0	0	0	0	0	0	0

3.2.1.3.2 Imposed Flow, Power and Concentrations Levels in a Branch

For flow type injection effects, it is also possible to assign the dissipation of a heat power level that heats up the air flow through the branch or which on the contrary cools it (negative power level).

Imposed flow, heat and concentration levels														
Effect type 4		Effect sub-type 2												
Scenario			Concentrations											
Time (s)	Flow (kg/s)	Power (MW)	Dryair	O	O2	Nitro	Helium	CO2	CO	H2O	Liq	SO2	HCN	HCL
0	20	5	0	0	0.3	0.7	0	0	0	0	0	0	0	0
60	20	10	0	0	0.3	0.7	0	0	0	0	0	0	0	0
180	20	15	0	0	0.3	0.7	0	0	0	0	0	0	0	0
240	20	20	0	0	0.3	0.7	0	0	0	0	0	0	0	0
2401	20	20	0	0	0.3	0.7	0	0	0	0	0	0	0	0
3000	20	20	0	0	0.3	0.7	0	0	0	0	0	0	0	0

3.2.1.3.3 Thermal Power Level Dissipated in a Branch

This effect is a basic development that served to develop realistic processing of a fire effect (thermal and toxic impact).

It comprises dissipating an amount of thermal power (a power scenario) in order to evaluate the thermal impact on the ventilation network. Note that the thermal impact is made primarily by combining four different physical phenomena in the ventilation network:

1. heating the gas flow in the chosen branch,

2. propagating the heated flow,

3. heat exchange between the fluid and the walls,

4. heat draw through when the tunnels slope.

The heating of the fluid that flows through the network is calculated at the point where the thermal power is injected based on a homogeneous mixture of the thermal power dissipated is expressed as:

$$\Delta T_{fl} = \frac{10^6 \, P_{thermal}}{Cp_{fl} Qm_{fl}}$$

where ΔT_{fl} is the homogeneous temperature difference generated immediately downstream from the fire, $P_{thermal}$ is the fire's convected thermal power, Cp_{fl} the calorific power of the fluid present upstream on the air side to the fire and Qm_{fl} the mass flow circulating within the tunnel where the fire is taking place.

This effect is obtained by using a simplified fire effect (with no toxic releases), the result being an effect file configured as follows:

Fire Effect type 5	Effect sub-type 2														
								Concentrations							
Time (s)	Power (MW)	Qm consumption of O2 (kg/s)	Qm combustion product (kg/s)	Dryair	O	O2	Nitro	Helium	CO2	CO	H2O - Vapor	Liq	SO2	HCN	HCL
0	0	0	0	0	0	0	0	0	0	0	0	0	0	0	0
60	5	0	0	0	0	0	0	0	0	0	0	0	0	0	0
120	10	0	0	0	0	0	0	0	0	0	0	0	0	0	0
180	15	0	0	0	0	0	0	0	0	0	0	0	0	0	0
240	15	0	0	0	0	0	0	0	0	0	0	0	0	0	0
300	15	0	0	0	0	0	0	0	0	0	0	0	0	0	0
360	15	0	0	0	0	0	0	0	0	0	0	0	0	0	0
4200	15	0	0	0	0	0	0	0	0	0	0	0	0	0	0

The other physical phenomena (propagation, heat exchange, thermal draw) are not considered to be specific effects that the user has to explicitly request, but are automatically taken into account by all of the branches of a network[3].

Note that propagation comprises evaluating from one time increment to another, a calculation of the movement of fluid masses within the entire ventilation network, allowing for the speed at which these masses move through each branch (a different speed depending on the location within the network and the density of the fluid at this point) as well as the mixtures that take place at the network nodes.

The thermal draw factor is taken directly from a simplification of the fluid mechanics equations and expresses the thrust produced by a surrounding fluid on a fluid mass with a different density from its own. In a sloping tunnel, this thrust value is expressed as:

$$\Delta P = g \left(\rho_2 z_2 - \rho_1 z_1 \right)$$

where ρ_i, and z_i respectively represent the density and altitude of point i, g being the gravity acceleration factor.

Concerning heat exchanges, these may be taken into account using varying levels of complexity, grouping all or part of the convection and radiation contributions as well as various aspects of exchange surface behavior: adiabatic, isotherm, constant or variable flow. The latter method making it possible to observe any heat storage and release phenomena affecting a wall.

These physical phenomena are handled by the general digital model and they are not developed in this report.

3.2.1.3.4 Imposed Concentrations within a Branch

This effect was not implemented for it is not really meaningful in reality. It can however be implemented and will probably serve for achieving 1D/3D coupling, so as to take into account, within the ventilation network, a fluid velocity from the three-dimensional area and with the concentration characteristics imposed by this area (refer to sub-section 3.5.3.). This can be written out in the form shown below:

---

[3] Note however that is possible to disable any allowance of thermal effects overall.

Imposed concentrations												
Effect type 2	Effect sub-type 1											
					Imposed concentrations							
								H2O -				
Time (s)	dryair	O	O2	Nitro	Helium	CO2	CO	Steam	Liq	SO2	HCN	HCL
0	0.00		0.00	0.23	0.77	0.00	0.00	0.00	0.00	0.00	0.00	0.00
6	0.00		0.00	0.00	0.00	0.66	0.03	0.31	0.00	0.00	0.00	0.00
12	0.00		0.00	0.00	0.00	0.66	0.03	0.31	0.00	0.00	0.00	0.00
18	0.00		0.00	0.00	0.00	0.66	0.03	0.31	0.00	0.00	0.00	0.00
24	0.00		0.00	0.00	0.00	0.66	0.03	0.31	0.00	0.00	0.00	0.00
30	0.00		0.00	0.00	0.00	0.66	0.03	0.31	0.00	0.00	0.00	0.00
36	0.00		0.00	0.00	0.00	0.66	0.03	0.31	0.00	0.00	0.00	0.00
42	0.00		0.00	0.00	0.00	0.66	0.03	0.31	0.00	0.00	0.00	0.00

Note that the developments currently under way could, with a few modifications to the tool, allow taking into account other gasses than those shown here.

3.2.1.3.5 Imposed Temperature in a Branch

This effect too, is not a very 'realistic' one. It represents an 'ideal' air-conditioning function able to, whatever the flow through a branch, impose a set temperature level.

This effect, not implemented to date, does not pose any particular problems and can be implemented in line with the same principles as those employed for imposing concentration values for a gaseous mixture. An example is provided below of what an effect of this type could look like.

Imposed temperature	
Effect type 2	Effect sub-type 0
Time (s)	Temperature (K)
0	288
60	300
61	400
2400	500
2401	500
3000	500

3.2.1.3.6 Simulating a Fire in a Branch

The fire effect was developed to allow a simplified but realistic evaluation of the thermal and toxic effects of a fire.

Consequently, for thermal effects we take into account a power level convected by the fire smoke, which when it mixes with the flow of fluid circulating through the branch where it is declared, has the effect of increasing the homogeneous temperature of the air flow downstream from the fire. Furthermore, the model developed can evaluate the toxic effects of mixing a flow of smoke produced by the fire along with a consumption of oxygen by the fire itself. By defining a scenario listing the various combustion products most present in a fire and possibly extremely toxic, it becomes possible to evaluate the toxic impact of the fire smoke in the ventilation network.

Time (s)	Power (MW)	Qm consumption O2	Qm combustion products	Dryair	O	O2	Nitro	Helium	CO2	CO	H2O - Vapor	Liq	SO2	HCN	HCL
									Concentrations						
0	0.00	0.00	0.00	0.00	0.00	0.23	0.77	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00
6	0.11	0.01	0.01	0.00	0.00	0.00	0.00	0.00	0.66	0.03	0.31	0.00	0.00	0.00	0.00
12	0.21	0.02	0.02	0.00	0.00	0.00	0.00	0.00	0.66	0.03	0.31	0.00	0.00	0.00	0.00
18	0.32	0.04	0.04	0.00	0.00	0.00	0.00	0.00	0.66	0.03	0.31	0.00	0.00	0.00	0.00
24	0.43	0.05	0.05	0.00	0.00	0.00	0.00	0.00	0.66	0.03	0.31	0.00	0.00	0.00	0.00
30	0.53	0.06	0.06	0.00	0.00	0.00	0.00	0.00	0.66	0.03	0.31	0.00	0.00	0.00	0.00
36	0.64	0.07	0.07	0.00	0.00	0.00	0.00	0.00	0.66	0.03	0.31	0.00	0.00	0.00	0.00
42	0.75	0.09	0.09	0.00	0.00	0.00	0.00	0.00	0.66	0.03	0.31	0.00	0.00	0.00	0.00

The model retained for dissipating the power into the fluid circulating in the branch is the same as the one retained in sub-section 4.2.1.3.3.

The model for mixing smoke is defined as follows, expressing a list of the products present downstream from the fire:

- For the concentration into a gas i, other than O_2 the result is:

$$[C_i]_{downstream} = \frac{Qm_{upstream}[C_i]_{upstream} + Qm_{combustion\ products}[C_i]_{combustion\ products}}{Qm_{upstream} + Qm_{combustion\ products} - Qm_{consump.O_2}}$$

- For the concentration in O_2 the result is:

$$[O_2]_{downstream} = \frac{Qm_{upstream}[O_2]_{upstream} - Qm_{consump.O_2}}{Qm_{upstream} + Qm_{combustion\ products} - Qm_{consump.O_2}}$$

At this stage, and to facilitate taking into account a fire in a ventilation network, the choice was made to neglect the change in mass flow of the fluid circulating upstream and downstream of the fire. We therefore estimate that the mass flow induced by a fire, i.e. the mass of combustion products reduced by the consumption of oxygen is low, compared to the flow of air that ventilates the fire (high smoke dilution).

In practice, fire smoke is often severely diluted (except in the case of under ventilated fires). This assumption based on a calculation of the toxicity of fires is not therefore much of a constraint, especially as there is little theoretical data on under ventilated fire so that to date it is impossible to apply a general model general simple enough to match with this calculation tool. We do however note the experimental work undertaken by Marlair *and al.*[12] covering the production of data on characteristics of under ventilated fires.

3.2.2 Processor

The processor lets the user run all of the main phases of a simulation of accidental effects in a grid network:

- calculating network connectivity as well as digitally preparing the resolution,
- defining physical, numerical and computer configuration parameters for the simulation,
- monitoring the calculations and storing results,

Automatically calculating network connectivity is a significant New-Vendis function compared with the previous versions. In this way, the user can retrieve the spatial coordinates of a branch network without needing to worry about connections between branches. Determining network connectivity 'by hand' can be extremely painstaking to perform and a significant source of error. Automatically calculating network connectivity reduces this error risk and virtually eliminates it when the connectivity result is viewed with the grid mechanism. Naturally an ultimate grid network check can also be performed by an in-depth analysis of the calculation results.

The connectivity calculation can also be performed in a number of ways, depending on whether network geometry was defined in full or only simplified by the user. In practice, the user can declare all of the branch network nodes, or just declare its branches (Figure 4—3).
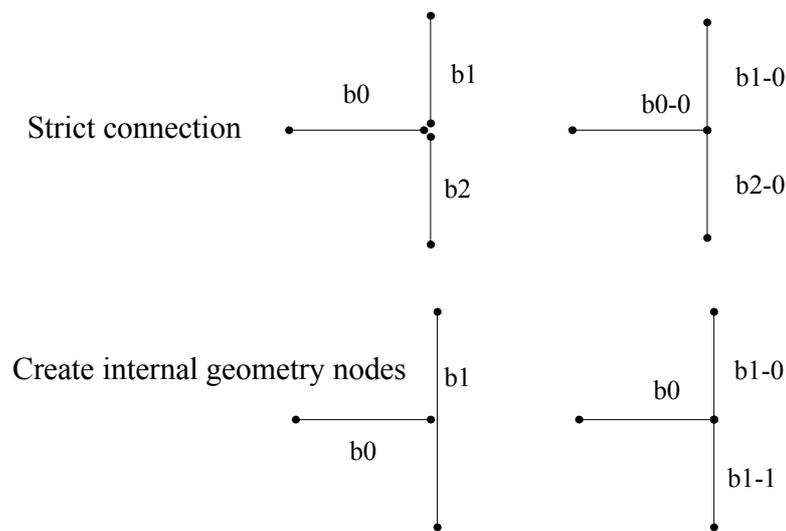
Figure 3—3: Presentation of the two methods available for describing a ventilation network. 'Strict connection' corresponds to when the user declares all of the points needed to describe the network nodes and 'Create internal geometry nodes' corresponds to when the user does not explicitly declare all network nodes.

Depending on the option chosen by the user, the connectivity calculation may or may not include a search for the missing node points. In all cases, the calculation assumes that two branches are connected if the node points verify the following relationship for each coordinate:

$$| Xi1 - Xi2 | < \text{connectivity length}$$

(with i varying from 1 to 3 for the three geometrical space dimensions)

When calculating the connections between network branches, the following dialog box is displayed:
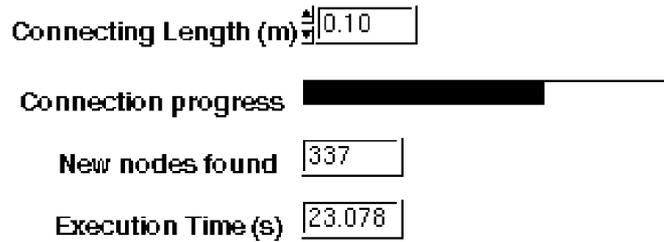
Figure 3—4: Dialog box showing connectivity calculation progress.

This makes it possible to observe progress during the connectivity calculation, which may take a while if a search is made for undeclared node points and if there are a large number of branches. Typically the connectivity calculation time varies from a few seconds to some twenty minutes for networks of many hundreds of branches.

After this step, the connections are grouped in the following data structure (Figure 4—5). All of the nodes, including any nodes that may be created, are numbered from 0. All of the branches, including the newly created branches are also numbered. For each node, the system then memorizes which branches are connected to this node and using which end. In the example below, we can see that node 0 has two branches connected to it. These are the branches numbered 4 and 0. Branch 4 is assigned index 0 as is branch 0. This means that these two branches are connected by their departure node (shown as *ndd*, whereas the arrival node is shown as *nda*). If the index number is not 0, this means that the corresponding branch is connected through its *nda*.

Figure 3—5: Data structure for network nodes.

The processor will also allow defining overall calculation physical parameters. The main ones mentioned are as follows:

- choice of pressure and temperature reference values at the initial instant, for the fluid contained in the network and the outside air,
- choice of the initial tunnel temperature level (mass),
- choice of the status equation used to calculate the density of the fluid flowing through the tunnels,
- choice of the type of thermal transfer (convection, radiation, conduction),
- choice of overall effects taken into account (wall friction model, allowing for thermal draw effects…).

As for numerical parameters, the processor allows setting a choice for calculation convergence (the method used for calculating the error and the tolerated error value) as well as a definition of simulation times and the number of steps in the calculation time.

The processor also allows saving results. This requires creating two files.

The first file is identified by the *.case'* suffix. It is used to save the calculation configuration as well as a full description of the test case, corresponding to the list of branches, the list of numerical points, the list of numerical branches, the initial mix characteristics, the overall parameters, the effects table and the atmosphere branches.

The second file is used to save calculation results. The components retained are chosen from a pre-selection menu as shown in Figure 4—6. It is therefore possible, for each time increment, to save the mass flow, the mass fractions that make up the air within the structure, resistance, density, load losses, relative pressure, air temperature, tunnel wall temperature, dynamic viscosity, the Reynolds number and the air speed.
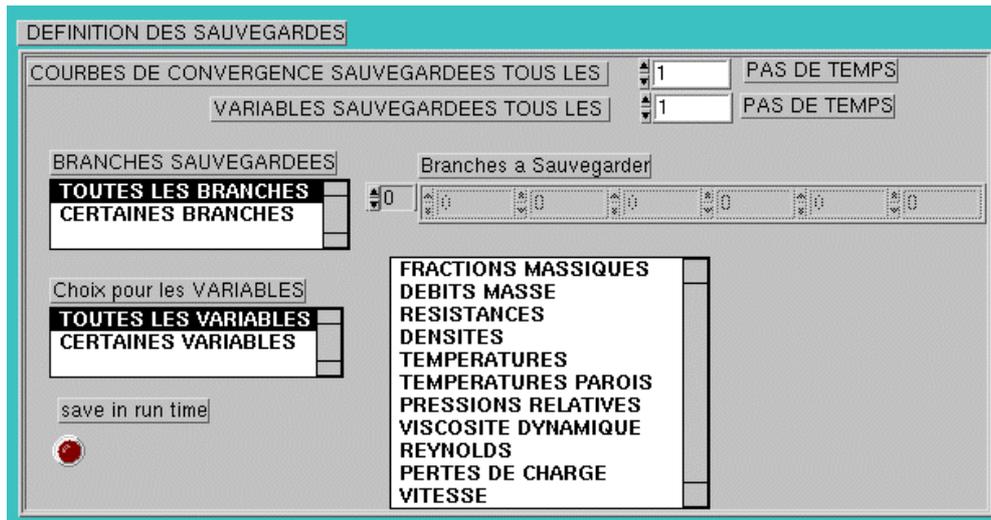
Figure 3—6: Presetting structures saved

Two save formats are available.

- The first format, with a *.save'* suffix, allows saving after the fact, once the calculation is complete. The preset data is then retained in memory, in full. This mode may needlessly fill up the computer's memory and consequently considerably reduce its performance.

- The second format, with a *'.rave'* suffix, allows performing a save during calculations. After every time increment, the preset elements are added to the save file opened at the start of calculations. This approach limits the amount of memory space taken up. However, reading back too must be performed iteratively which implies significant actions in memory which may cause retrieval time to be longer.

Retrieving results depends on the storage format used, but the type of post-processing that is possible later does not depend on the save format used.

Simulation will then be calculated describing the characteristic equations of each branch. An analogy is then performed with electrical networks by writing the node law for flows and the grid law for load losses. To these equations are added the following additional terms characterizing the specific effects affecting them as well as any local refinements that may be made to the calculation case.

The grid network theory will then allow determining any unknowns and choose independent equations from among all of those which are representative of network balance. To resolve these equations, we apply the Hardy Cross numerical relaxation method. [9]. The numerical methods will not be developed in this summary report.

3.2.3 Post-processor

The post-processor is in fact made up of a number of tools for analyzing results.

The main options are:

- An overall view of the velocities reached by color level, possibly including a marking of flow direction.
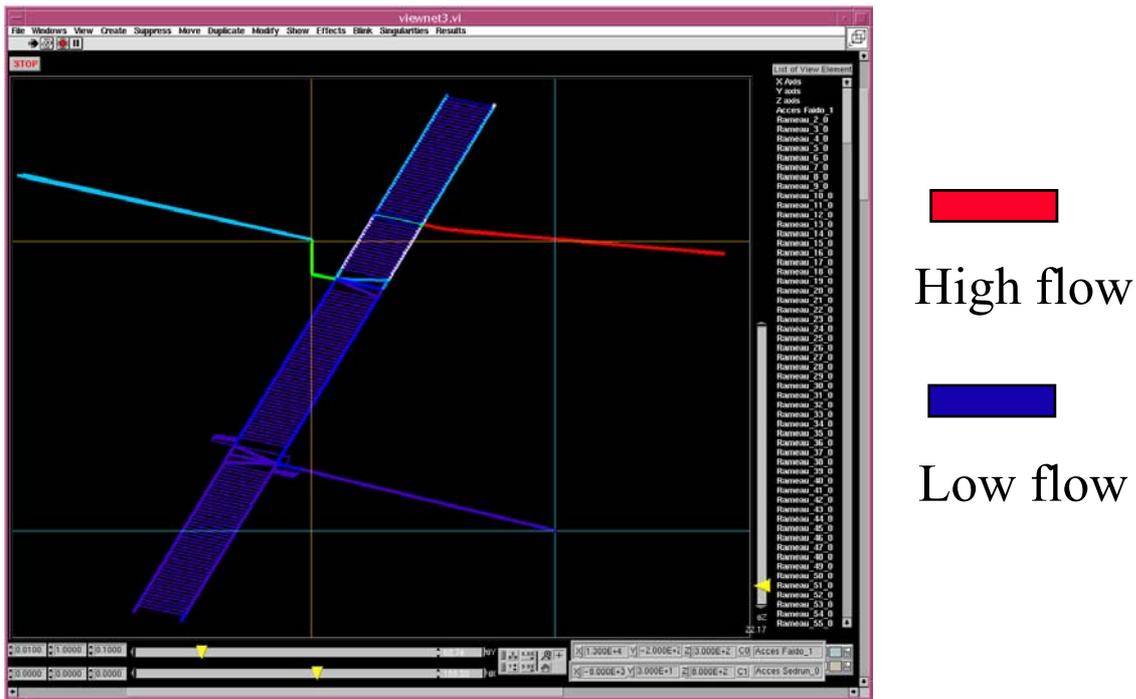
Figure 3—7: Displaying the flows through a network by displaying color levels.

The result analysis may also cover the flow direction within the various conduits.
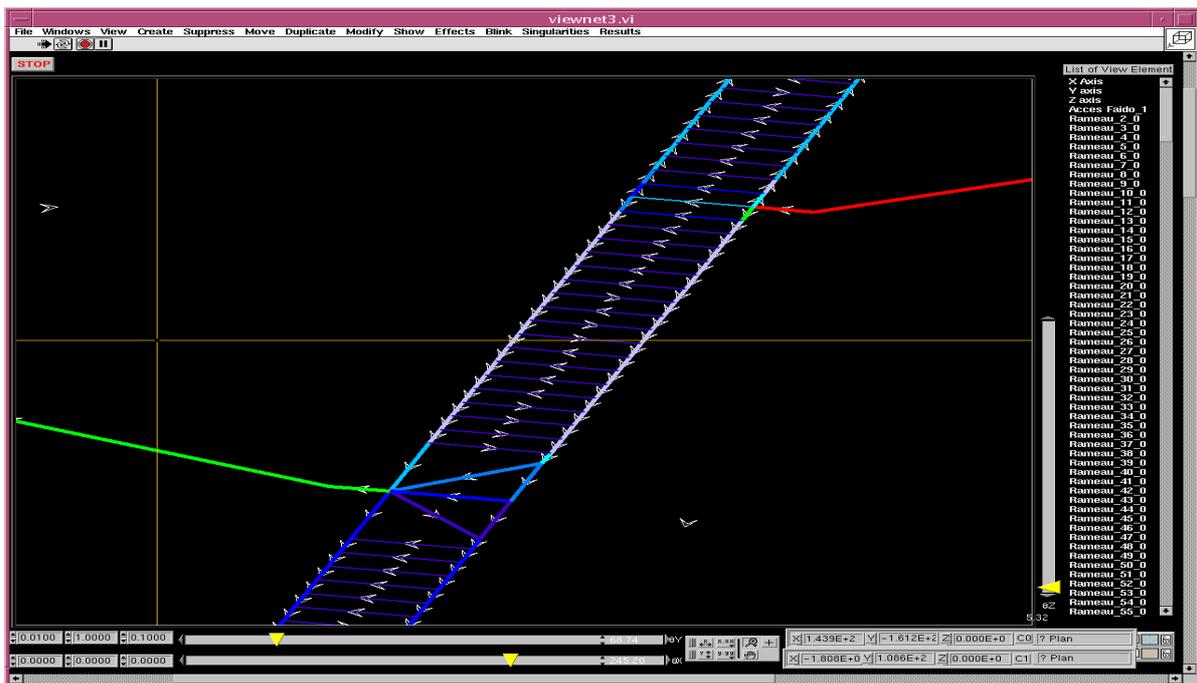
Figure 3—8: Displaying color levels and flow directions.

- Viewing the different flow rate parameters along a given branch or, a variation over time of the flow rate parameters at a given location on the network can be obtained by generating text files that are usable with a spreadsheet (e.g. Microsoft Excel),
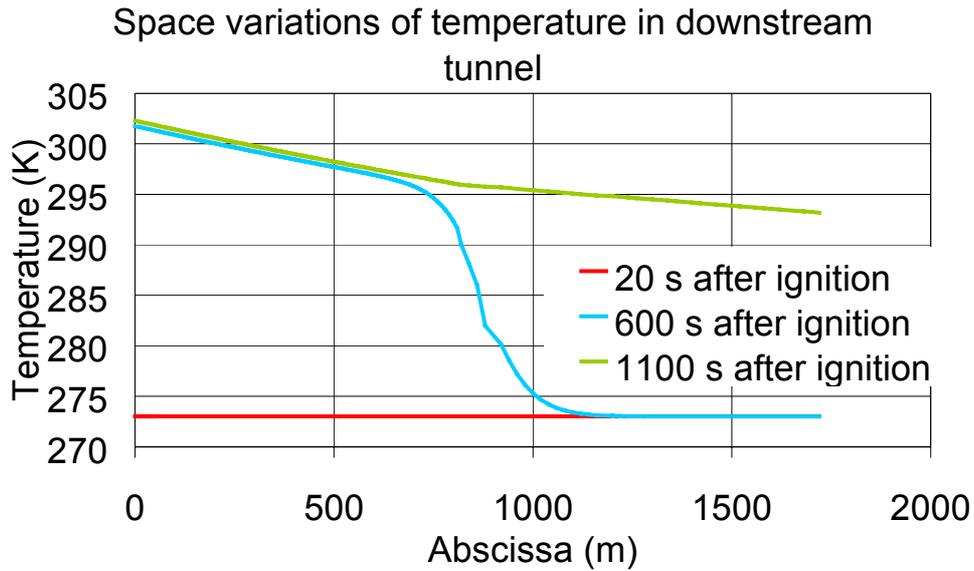
Figure 3—9: Example of temperature changes within a branch at different times.
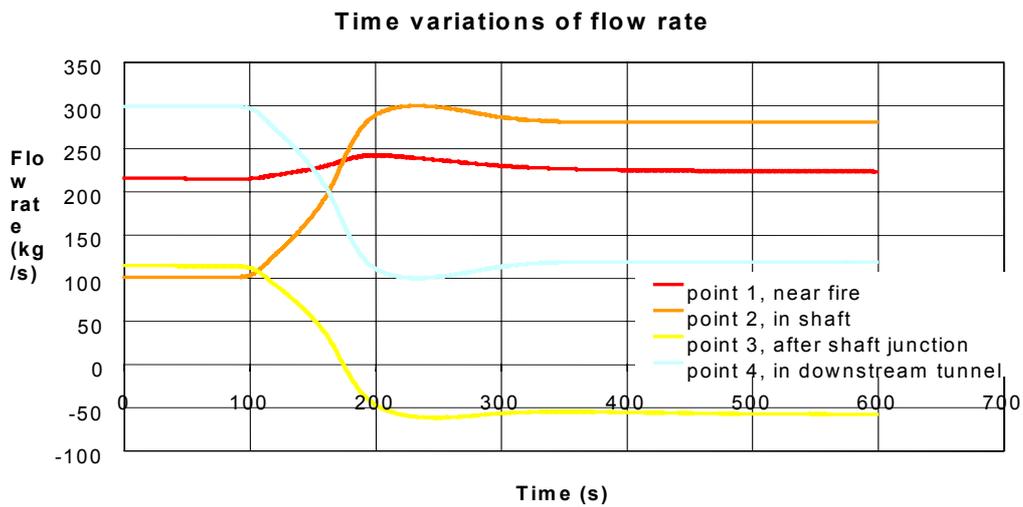
Figure 3—10: Example of time variations in flow rates in different branches of a network in a departure and arrival state.

- Viewing different flow rate parameters in the form of space-time diagrams (for a fictitious example of CO_2 dispersion present in a tunnel fire).

Cas :   Scenario d'incendie factice

Lieu :

Propagation des fumees dans un tunnel

Variable :

CO2
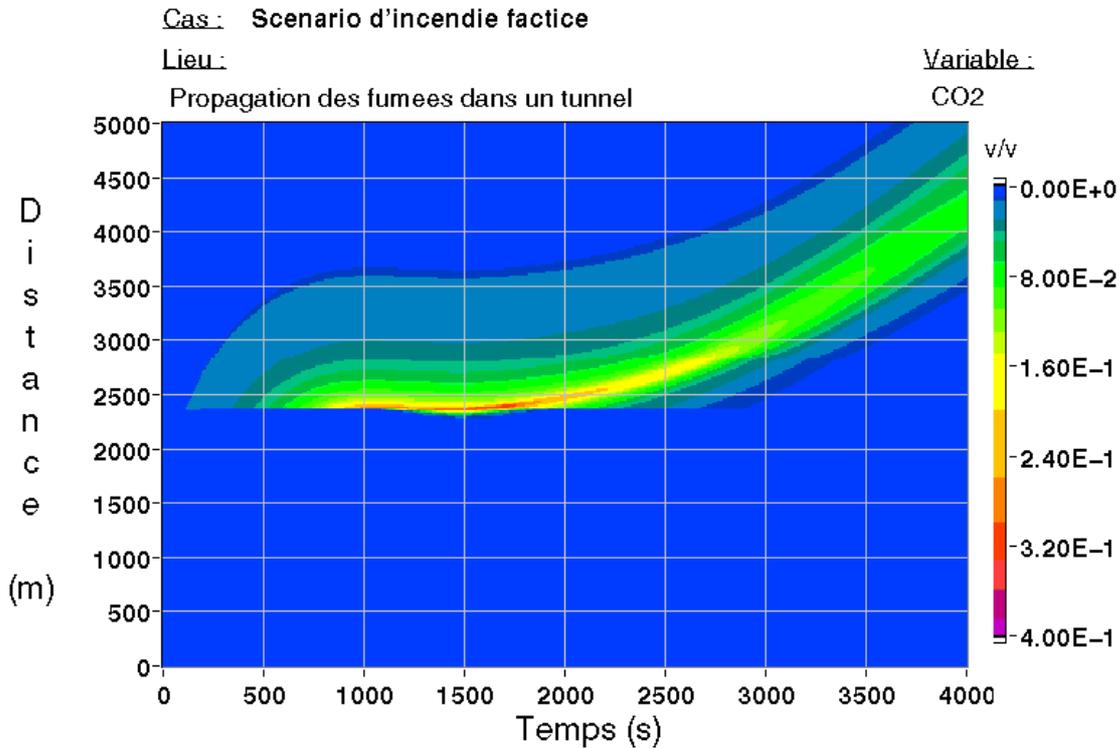
Figure 3—11: Example of a Space/Time diagram showing smoke propagation in a network tunnel.

3.3   1D/3D COUPLING

In this study, the coupling implemented requires a 1D model (New-Vendis) along with a 3D model (Phoenics). The coupling logical structure remains open however to more general kinds of operation. It has therefore been developed to all coupling New-Vendis with other models.

3.3.1 Functional Architecture

The coupling between a number of numerical models can be implemented in a number of different ways.

The first, and certainly the most natural solution, is to merge the different models into one. Coupling is then mathematically resolved during the mathematical resolution phase. The main drawback to this technique is the need to develop all of the models to be coupled. This also implies considerable difficulties when it comes to possible changes to the coupling system. The 1D/1D coupling presented in sub-section 3.5.1. is built in this way, although it remains relatively easy to externalize.

The other solutions imply making the calculation components that are specific to each model communicate. This means running a 1D calculation code with, for example, a 3D calculation code. If the two models simulate non-stationary principles, then calculation synchronization, in terms of execution and de-discrete processing in time, become indispensable.

The coupling diagrams below are possible. They use a master/slave model as shown in Figure 4—12 and an equitable distribution model also called a distributed model (Figure 4—13).
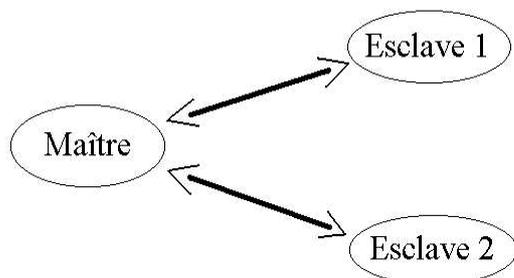
Figure 3—12: Master/slave model   Figure 3—13: Distributed model
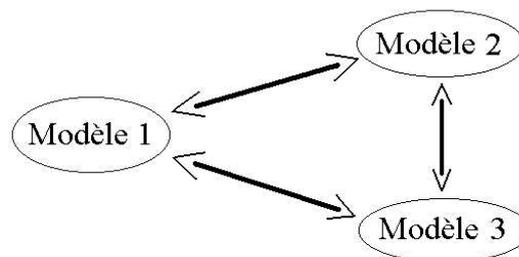
The model distributed model assigns an identical role to each of the models and consequently to each of the calculation codes that implement it. The links between each entity are entirely symmetrical. The main component of this architecture is the concept of links between models. Coupling will then be expressed like a digital and computer system entity that homogeneously links a set of models.

The master/slave model considers that one of the components has responsibility for coupling. It fully controls and manages the other coupled elements as well as extracting data from them.

Taking our context, the distinction between a global and local model is a natural one. The master/slave type structure appears to be best suited for achieving coupling. In this diagram, the master is therefore New-Vendis (the global model) while the slaves are the different codes which contain the models used to calculate the local phenomena. This choice allows centralizing coupling management in New-Vendis and thereby restrict the specific intervention required on external code. This also allows retaining coherence in the coupling concept, in New-Vendis, as the 1D/1D coupling is managed in the same way under the control of the Hardy Cross method, the heart of New-Vendis.

The main difference between 1D/1D coupling and 1D/other model coupling is that the other model is externalized.

3.3.2 Implementation in New-Vendis

The 3D model is used to simulate a flow through a part of the network. This resolution must not be so redundant nor so concurrent with the single-dimensional model. This calls for a single representation of each of the parts of the network.

Note that a certain degree of overlap in the models may sometimes be useful to resolving digital stability problems.

3.3.2.1 Creating an Equivalent Network

The area calculated by the 3D model is no longer a part of the 1D network, which means that it is open at all of the coupling points. However, to allow balancing to be calculated, the network needs to be closed. To do this, the concept of an equivalent network is introduced.

Figure 3—14: Diagram view of a coupling

Figure 3—15: View of the equivalent 1D network

As shown in Figure 4—14 and Figure 4—15, the idea is to replace the 3D calculation area with an equivalent 1D network. The geometrical view of this network is obtained by prolonging all of the branches connected to the 3D area to the central node. From a practical point of view, the center of gravity of the ends of the branches allows this central point to be determined easily. The latter's position can however be chosen in any way as the characteristics of these new branches are entirely determined by the coupling function without taking into account their geometrical characteristics.

The declaration of the coupling areas as well as the creation of the equivalent network are performed using the New-Vendis pre-processor. This is used to possibly declare a number of coupling areas. A coupling area designates a geographical area of the tunnel network for which a code that is external to New-Vendis is used. This area is therefore described by its identifier, the external calculation identifier, the type of coupling made and by the list of branches entering the coupling area. This structure is used to declare a number of coupling areas. Once the coupling area is fully declared, a tool is used to build the area's central node and to assign each of the newly created network branches a dynamic coupling effect that matches the kind of coupling chosen.

The same effect notion is reused here as was presented in sub-section 3.5.2.1.1.

Figure 4—16 and Figure 4—17 show the result after building the equivalent network using the pre-processor.

Figure 3—16:

Figure 3—17:

3.3.2.2 Role of Dynamic Coupling Effects

The equivalent network makes it possible to retain a closed network for resolving a Hardy-Cross. However, geometrical closure comes with a set of parameters and effects for each new digital branch created. As these branches are nothing but a digital artifact, the resolution that are given must absolutely only be controlled by the coupling. The overall effects like the momentum load losses are therefore meaningless for these branches. To this extent, branch parameters are set in such a way that these effects are cancelled out. The pressure variation along these branches is caused solely by the coupling effects that are representative of local refinement.

From the coupling area point of view, the drafts blowing through the branches are either incoming or outgoing. The coupling effects relating to each equivalent network blanch are used to give the 1D network the right information or take it from it. However the split between the incoming or outgoing branch may change over time, in line with changes to network balancing. The coupling effects therefore take different forms over time, these are called dynamic effects.

The basic effects required such as imposing a flow, a temperature, a heat power level or concentrations and pressure jumps are therefore implemented, for each time increment and for each coupling area.

3.3.3 Some Thoughts on the Coupling with Phoenics

The coupling between New-Vendis and Phoenics (CDF 3D calculation code) was more profound from a conceptual point of view without however actually being implemented and tested.

Phoenics is the 3D calculation code built to simulate the running off of fluids that are only slightly or not at all compressible. It takes into account heat transfer phenomena such as radiating transfers and heat conduction through solids. The calculations can be made for turbulent run off, using one of the turbulence models proposed. System closures are mainly of the first order, with the use of two conservation equations that are specific to the turbulence model. This is, for example, for the k-ε model, the turbulence's kinetic energy and turbulence dissipation.

The diagram used for converting into discrete data is a "finite volume", "cell centered" type one, that allows either the collocation of all unknowns or a shifted grid pattern for some of them. The resolution method used is a SIMPLE type, i.e. a predictor/corrector type approach is applied to pressure.

For this type of calculation code, the inlet edge conditions are generally imposed through weak formulation for all of the variables except for pressure. The latter is generally the only variable that is set for outlet edge conditions. This configuration of limit conditions makes it possible to leave pressure waves free to establish themselves. This is because the flow rates considered here are subsonic ones, so consequently there is a pressure wave that propagates in the outgoing flow until the edges of the calculation range. If the pressure level is set for all edges, the wave cannot 'escape' freely and the calculated pressure (if the resolution method is able to converge) will then present purely digital oscillations that may affect virtually all of the calculation field.

The quantities required for 3D calculations are therefore as follows:

- input: the three components of the speed, temperature (or enthalpy), the fractions of each chemical species calculated, the kinetic energy of the turbulence and its dissipation (in cases where the k-ε model is used),

- output: pressure.

These quantities must therefore be provide by New-Vendis.

From the 1D model point of view, the data from the 3D area must be communicated. This includes the temperature, gas mass fractions and pressure. As the momentum load losses are null in the coupling branches, the pressure can only be imposed by a pressure level jump.

The temperature and mass fractions provided by the 3D code are average values in each area exit section. This means that the 3D model can be attached to a 1D model with constant quantities for each tunnel section and that are conservative. As for pressure, homogenized values are obtained identically. However, for the Hardy-Cross iteration, it is necessary to have pressure jumps between the equivalent network central node and the connection branches to the 1D network. These pressure jumps are obtained using a difference from the average homogeneous pressure levels on the inlets and outlets to the 3D area.

The 3D code's input data raises a different problems. At this point it is necessary to build surface information (the size of the 3D entry edge) from scalar data (the size of the quantities exiting the 1D model). A number of solutions are possible. The first one comprises uniformly applying the scalar value over the surface. This solution implies a non physical triggering of the development of the limit layer. Another solution is to use an adimensioned profile in such a way that its average is the unit. Then all that is needed is to multiply the profile by the adimensioning scalar. The complexity is then lies in determining realistic input and output profiles. This technique does however allow the use of edge conditions that are less difficult for resolving the 3D model, especially for the turbulence variables. This was used successfully as part of the fine scale modeling of gaseous sprays, [13]. As appropriate, the precision brought by the use of more or less relevant profiles depends on the singularities to be shown (sudden narrowing, widening, injection from a tube or directly from a tank…).

The role of the calculation done using Phoenics comes down to determining the values assigned to the dynamic coupling effects during each time increment. It therefore seems possible to take into account the consequences of local three-dimensional effects as well as the influence of network balancing on the local phenomena.

3.3.4 Computer Factors

From a computer point of view, coupling can be achieved in a number of different ways.

One solution is to integrate one calculation code into the other. To do this, the imported calculation codes must be accessible in computer library from[4].

A second solution is to use a different process[5] for each calculation code. Communication can then take place in a number of different ways. Reading and writing to files is certainly the simplest technique although it is difficult to automate and especially, not very efficient. Direct communications between different processes is possible using techniques such as shared memory[6], semaphores[7], pipes[8] or sockets[9]. However, it is only by using sockets that it becomes possible to communicate between processes run on different computers.

Reseau=Network
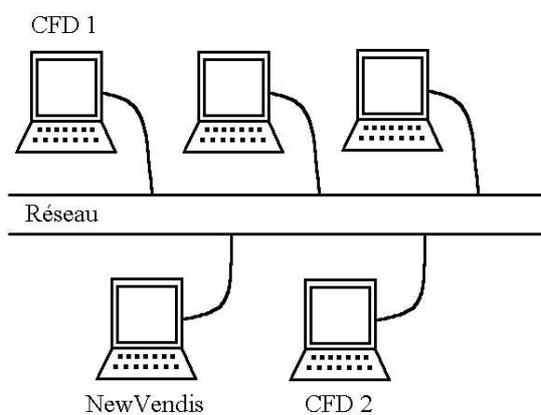
Figure 3—18: Diagram showing a possible coupled calculation configuration

The chosen architecture is therefore based on the use of techniques that allow process coupling on different computers as illustrated in Figure 4—18. The goal is therefore to base the work on the software tools implemented for professional and general public computer networks such as the TCP/IP communication protocol. This choice fully meets the requirement for portability. Choosing LabView for developing New-Vendis is also justified by the large degree of portability offered by this product. This difficulty is entirely managed by the vendor in the case of Phoenics. Network communications thereby allow retaining significant flexibility from this point of view.

---

[4] File containing the executable form of a set of procedures, functions or objects

[5] The process concept refers to a memory location with a binary code and its execution by the processor. This represents the set of elements relating to program execution.

[6] A number of processes share the same physical computer memory segment. This technique requires full control over process coordination.

[7] This is a binary form of shared memory often used for synchronizing processes.

[8] The pipes may be represented as communication pipes between processes. Their complete management is handled by UNIX type operating systems.

[9] The sockets are virtual connectors that are accessible by the processes that created it as well as by all other processes authorized from a computer. These virtual connections are often used in high level network technologies like the web.

Another important criteria in this kind of choice relates to the ability to share computer resources. Processes operating on separate machines that do not share resources (processor, memory, operating system). It is therefore entirely possible to use a high powered processor for the 3D code and a more modest computer for New-Vendis.

3.3.4.1 Communication Strategy

New-Vendis includes procedures that allow it to manage two-way communications with a number of other processors. The way the calculation system operates to perform a coupled calculation can be broken down as follows:

- Calculation preparation:
    - building the 1D parts (normal grid mechanism usage),
    - defining coupling areas,
    - building equivalent networks,

- Calculation process:
    - starting New-Vendis, waiting in server mode until all of the clients have connected,
    - simulation loop, New-Vendis primarily awaits responses from clients,
    - sending the end of calculation order to clients and shutting down simulation.

- Post-processing performed with tools that are specific to each model.

The work performed in the New-Vendis pre-processor enables defining the coupling performed. Each coupling area is assigned to a client, i.e. a process. The area name is the identifier used during the connection request. The type of coupling is also declared when the area is created. It designates the effect strategy assigned the coupling.

During the calculation phase, New-Vendis always remains on standby for requests from clients. The communication protocol lets New-Vendis determiner whether the client wishes to provide data or receive it. Consequently, the two-way connection management is performed by New-Vendis at the clients request. Synchronization, however is entirely managed by New-Vendis as the client continues to await a response until they have received the requested data.

3.3.4.2 Connection Protocol

During the connection phase, New-Vendis awaits connection via the preset socket port that is known to all of the clients. Each time connection is attempted, authorization is given for a preset length of time during which the client has to identify themselves by given the server their name. If the name is not recognized, the connection is cancelled by the server.

In server mode, the system continues to await connection until all of the clients expected (declared in the pre-processor) have connected. Each client therefore has a separate connection to the same socket port opened by New-Vendis.

3.3.4.3 Communication Protocol

3.3.4.3.1 High Level Protocol

The protocol has three commands. They are expressed in the form of the following character strings:

- « -> » the client will send data,

- « <- » the client awaits data,

- « stop » the other party indicates they are stopping.

The announcement that data is being sent is immediately followed by the data itself. New-Vendis then switches to read mode on the socket port.

For the client, a data request is followed by waiting in read mode and for New-Vendis by sending the data as soon as it is available.

The stop signal makes it possible to retain the updates for the connected clients but also to indicate to them the end of their activity.

3.3.4.3.2 Low Level Protocol

Although the TCP/IP communications used are secure ones, a low level protocol is implemented to guarantee the integrity of the data sent.

All of the data sent is communicated in character string form. This technique is used to guarantee the legibility of these messages on all kinds of computers. This is because the 128 ASCII code characters are standard and comprise all of the alphanumeric characters. This does however imply changing type before writing and after reading for all non-alphabetical messages.

To guarantee that in read mode, all of the data sent to the process has been read, communications are built as two messages. The first one with a fixed four character length gives the length of the useful message. The next one contains the useful data and exactly matches the stated dimensions. Consequently the reading process knows when it can handle and use the data received.

3.3.4.3.3 Communication Libraries

Two communication libraries were therefore developed in C language. The first one implements the low level protocol, as well as the procedures for opening and closing connection. The second one offers a higher level interface for FORTRAN. It can therefore be used directly in a FORTRAN source code and implements the high level protocol.

Both of these libraries were tested and validated on a purely computer based coupling between New-Vendis and Phoenics. This was used to show the reliability of the technological choices made.

4. SUMMARY PRESENTATION OF EXPERIMENTAL DEVELOPMENTS

As described in sub-section 3.3, the choice was made to develop the experimental requirements relating to this project with the aim of allowing wider use in the future, of the resources implemented for this project, and therefore the small scale simulation of accidents.

Consequently an entirely new, small scale test laboratory was built as part of this project, integrating the requirements for data validation that were specific to the project.

The laboratory created is described in brief, along with the test configuration implemented for validating the 1D/3D coupling and the main goals and test results obtained to date.

4.1 MODEL LABORATORY

The laboratory (refer to Diagram 4—1 and Photo 4—1) were the model is located is 21 meters long, 4 meters wide and 3.5 meters high. It is ventilated by a controlled air feed, split between three air inlets, each able to deliver 1,000 cubic meters per hour. The air flow in is also filtered and temperature regulated, to extend the periods during which the model can be used into cold seasons.

Four extraction stacks, 250 mm in diameter, are provided for evacuating gas and smoke (non toxic and non-inflammable) generated during the tests to simulate accident effects as well as excess fresh air, not required for ventilating the model. By controlling stack opening, it is possible to act on the pressure within the facility.

The reason for having facility ventilation suited to the experimental applications comes from the fact that this allows simulating long duration accidental effects without polluting the facility and therefore without altering the reference conditions under which the measurements were performed.
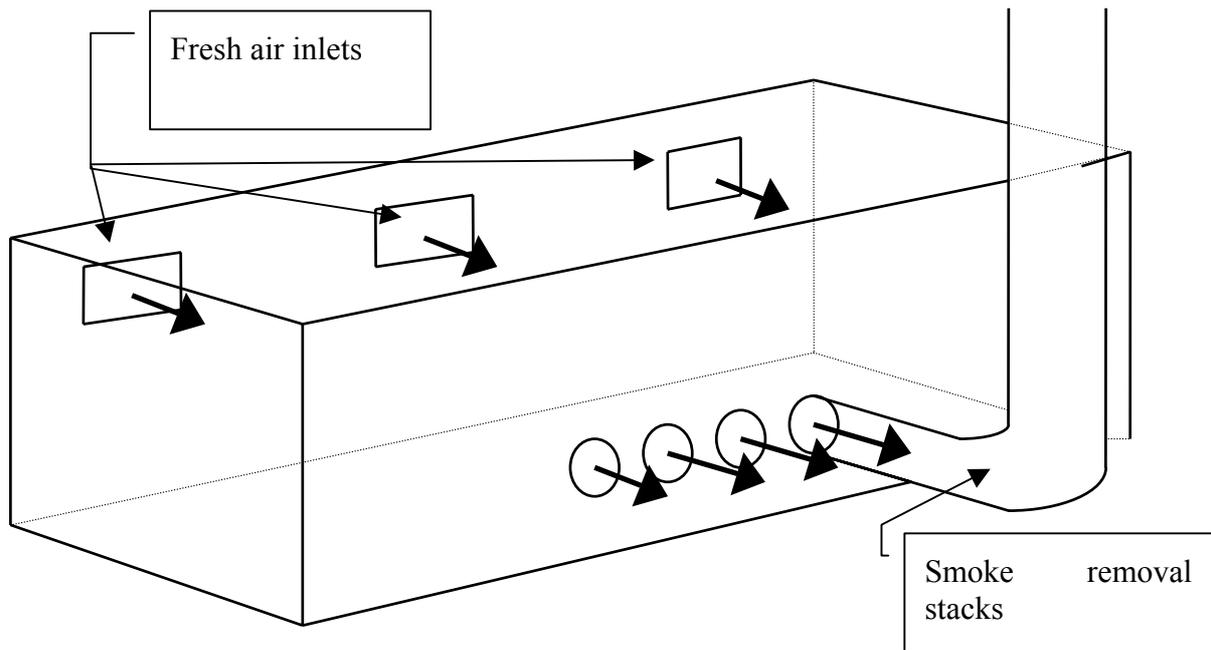
Diagram 4—1: Model laboratory ventilation.

Photo 4—1: Model facility overview showing the model base and the three-dimensional dispersion volume made up of see-through panels.

The model base is a 15 meter long, 1.5 meter wide, 1.5 meter high platform. Its long length is justified by the fact that it is designed to also accept 'tunnel' type model configuration. Furthermore, it can be inclined to the right or to the left around an axis located at ground level in order to simulate a maximum slope of +/- 10%.

Mounted on six casters, two technicians can move it around the facility and it can also be moved out of the facility.

To the model base are added metal arcs (refer to Diagram 4—2) which are intended to supporter a ventilation network.

4.2 THE EXPERIMENTAL MECHANISM FOR VALIDATING 1D/3D COUPLING

The experimental mechanism implemented will serve to validate the digital 1D/3D coupling model, the development of which started with this project. As the project's ultimate goal is to allow coupling of a 3D dispersion calculation code with the calculations made in a tunnel network (1D code), the experimental tool takes into account two separate interconnected run off zones:

- a zone formed by a parallelepiped that is representative of a cell, and
- a zone formed by a ventilation network.

This system is installed in the specifically set up facility area.

The three-dimensional volume is visible in Photo 1 in its 'cell' configuration. It is also visible in Diagram 2 which provides an overview of the ventilation network.

The materials that make up the three-dimensional model volume were chosen to obtain good levels of visual quality and so to be able to see a quality view of smoke behavior (e.g. a Laser plan) as well as to allow anemometric measurements using optical methods (e.g. LDA, PIV), in the future.

A large degree of three-dimensional volume modularity was one of the design goals. Consequently using the components visible in Photo 1, it is possible to simulate both a reduced scale area or a road tunnel that in reality would measure 300 meters in length. Other configurations can also be envisaged in the future, such as underground parking facilities or warehouses.

To recreate the volumes of the different test configurations that are possible, a series of transparent Plexiglas panels has been created. These panels are 30 cm wide and designed to slot into each other. The height formed in this way by a wall a number of panels high will be a multiple of 30 cm. It is however possible to precisely set a defined height by using other panels made up specially.

The vertical metal frames are made up of pillars the same size as the panels, mounted end to end. The frames and the walls slot together at will to achieve the desired cell length.

All of the Plexiglas panel mounting components are placed outside of the cell, so that the internal flows do not meet any obstacles. It therefore becomes possible to generate flows with well control limit conditions, something that is necessary for interpreting and using measurement results.

4.3 THE TEST MODEL

The ventilation network (refer to Diagram 4—2) that is linked to the three-dimensional volume is designed so as to be usable for the various three-dimensional volume configurations. The set up shown in Diagram 4—2 is the one that was made to reproduce as closely as possible the flows away from the different actual fire tests that were performed in the INERIS 80 cu. meter test cell (refer to Photo 4—2). This cell, also used at INERIS for other fire research projects, is used to set up real fires in a confined space under different ventilation conditions.

The general design parameters for the model's ventilation network comprise:

- the ability to reproduce in small scale, the fires studied in the INERIS 80 cu. meter cell,

- its ability to adapt to a number of operating conditions with differing flow levels and air and helium[10] content levels,

- a branch for re-circulating the air-smoke mixture used to observe the phenomena that are found in real-life when the ventilation system is equipped for fresh air recycling,

Three-dimensional volume (in the 80 cu. meter cell configuration)

Ventilation network support frame

Ventilation network
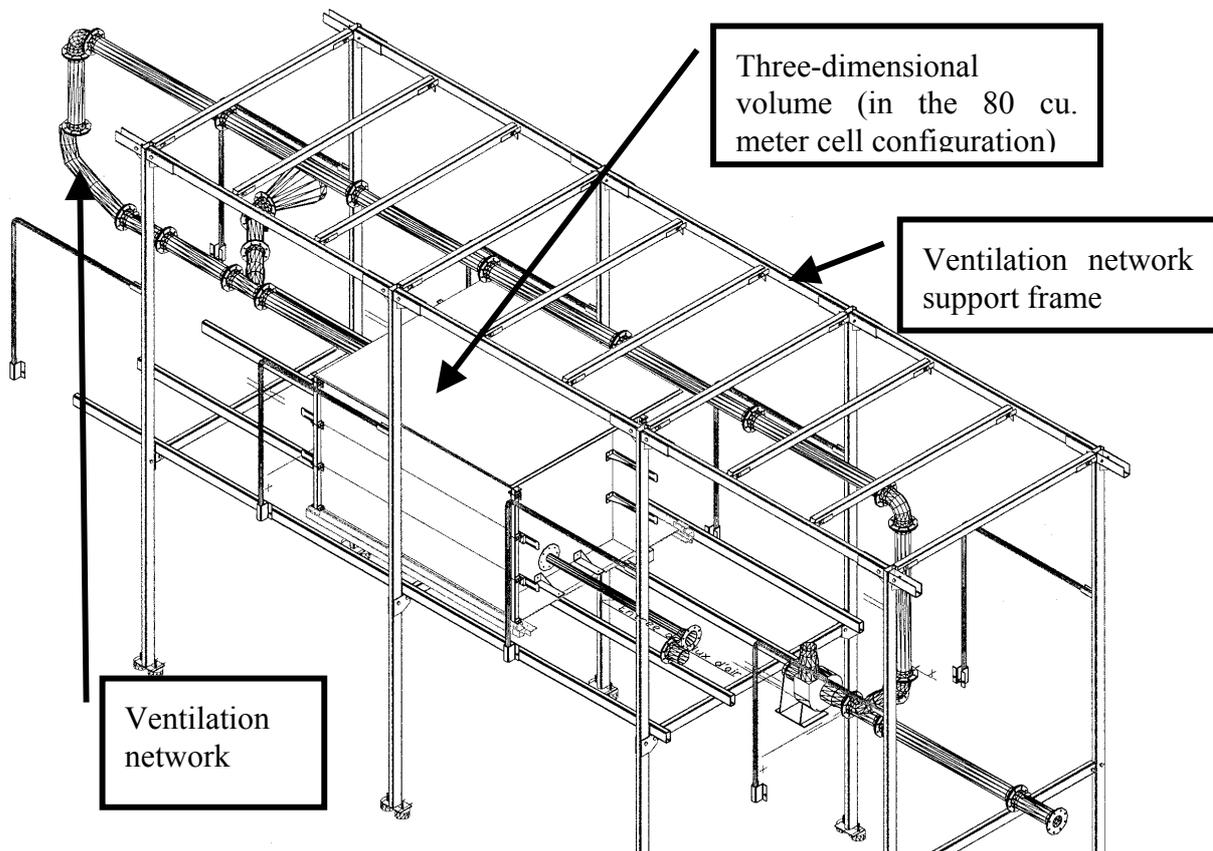
Diagram 4—2: Three-dimensional volume used for studying dispersion linked to the ventilation network

---

[10] Remember, fire smoke is simulated by Air-Helium mixtures.

- This re-circulation function also allows characterizing simulated smoke mixtures generating transitory phenomena for un-balancing the network, induced by the source of the fire itself or by opening and closing flaps,
- Its modularity for simulating other study cases (tunnels, underground parking facilities, warehouses...) impose the use of ventilation network components (pipes, elbows, registers) that are easy to disassemble and reusable.

Photo 4—2: Overview of the test dedicated to 1D/3D coupling ventilation.

4.4   SIMULATING ACCIDENTAL SOURCES (FIRE, GAS LEAK)

A smoke generation system was developed (refer to Diagram 4—3). This system allows generating smoke to simulate the hot smoke of a real fire. The generation of simulated smoke is based on the thermal/mass similitude of smoke, i.e. the variations in density relating to an increase in temperature in the case of fire smoke, are simulated by an air-helium mixture with a suitable helium concentration. Simulated smoke generation also comprises adding a 'show smoke' type aerosol spray so as to view the trajectories followed by the smoke escaping. The seeding on the escaping flows which is required for applying the speed measurement technique using a Laser Doppler effect Anemometer (LDA) will initially be performed using this set up.
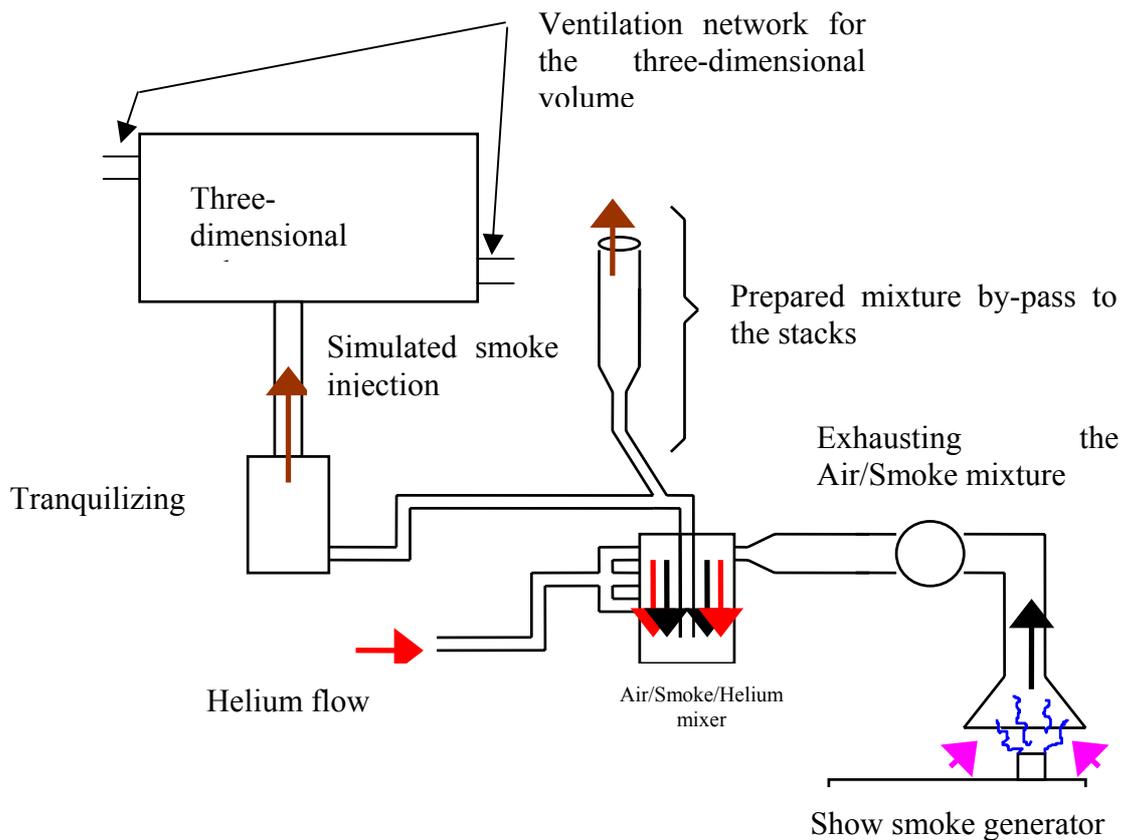
Diagram 4—3: Simulated fire smoke injection system.

4.5   PRINCIPLES OF TESTS CARRIED OUT

The principles adopted for the tests carried out to date were strongly influenced by the reduced amount of time available at the end of the project. Consequently, they were restricted to running only one full 1D/3D coupling validation experiment.

In the end it was possible to perform a set of measurements and displays using a smoke escape pattern including two end of test protocol variants. The simulated escape pattern corresponds to a cell with a fresh air ventilation flow that is constant throughout the test an in which the presence of fire is simulated by an air-helium mixture. The chosen air flow levels and settings are not intended to represent a specific fire but rather were chosen to obtain stratified smoke layers within the cell and hence, cause significant heterogeneity between one part and another within the three-dimensional area (the cell). It is important to be able to correctly calculate this heterogeneity and allow for it in an overall escape flow that includes a 1D zone and a 3D zone.

The chronological pictures of the sequence (Photo 4—3 to Photo 4—6) collected during one of these tests is shown overleaf.
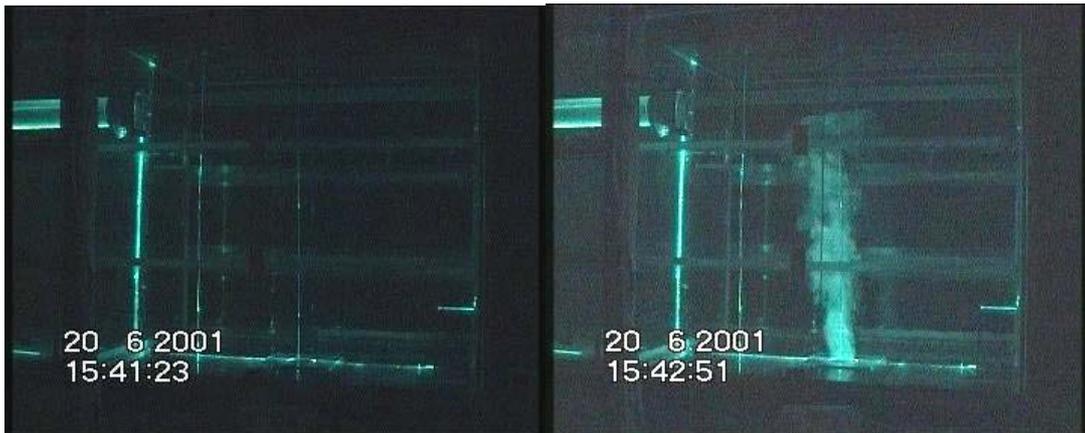
Photo 4—3: Reference display prior to testing (left) and the start of 'fire' type accidental event source injection (right).
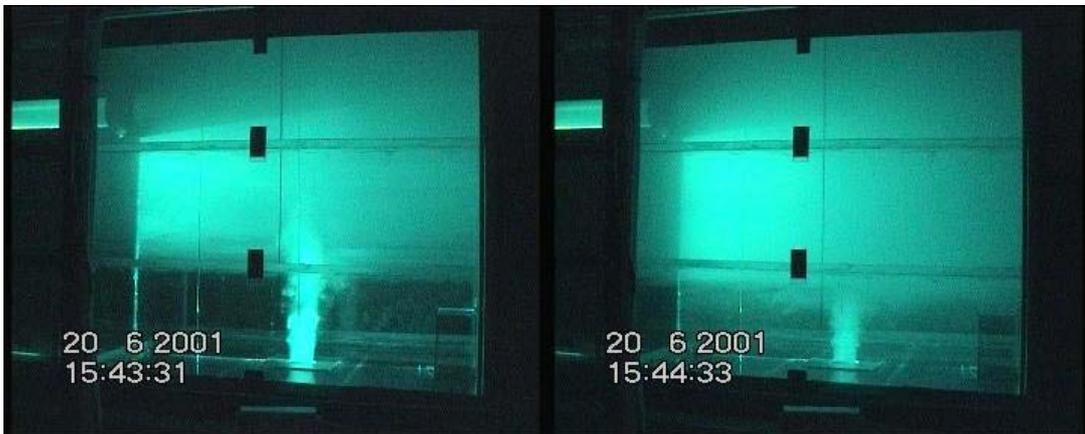
Photo 4—4: Evolution of stratification.

Photo 4—5: Stratification during the stationary test phase (left) and during cell ventilation after shutting down the source (right) implying de-stratification.
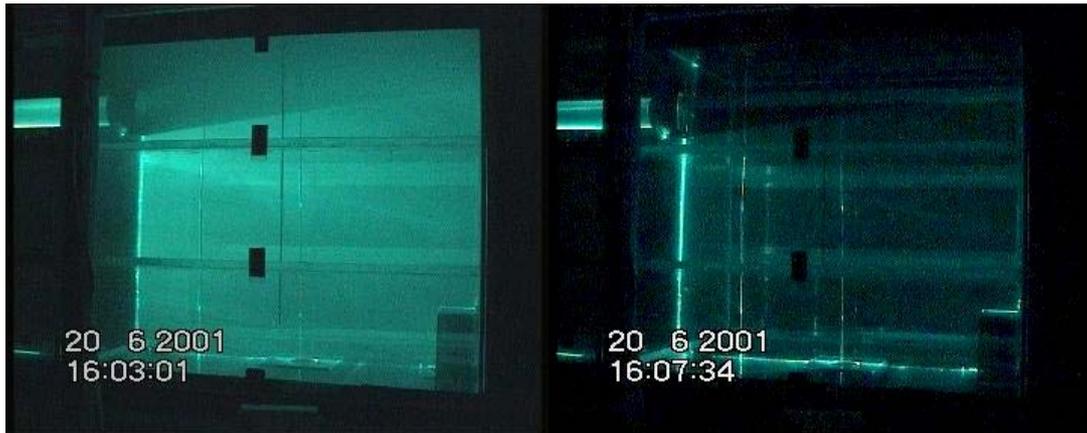
Photo 4—6: Cell ventilation after shutting down the source (left) and reference display after testing (right).

The knowledge of a single test regime is certainly not enough to fully validate the 1D/3D coupling. However, the data collected on the test regime described here is complete enough, both from the point of view of precise knowledge of the aeraulic network (1D) set up around the three-dimensional (3D) cell and as regards the 3D cell itself.

This is because, for a test regime, we obtained a full description of the flows, pressure levels and concentrations relating to the 1D network as well as fine description fine of the distribution of the helium concentrations (representative of the accident source) within the three-dimensional area. Furthermore, these measurements, taken for this test regime, cover both the stationary and non-stationary phases of the regime.

As a general rule, the tests undertaken were performed with the aim of ensuring the quality and the usefulness of the measurements obtained. Consequently, repeatability tests were undertaken a number of times and all of the measurements were validated by COFRAC connected calibration. Tests also covered checking the influence of the display method on the measurements. These tests showed that the exhaust flow display system did not influence the exhaust flow itself and therefore behaved like a tracer that was representative of the simulated fire smoke content.

5. SUMMARY PRESENTATION OF DIGITAL AND EXPERIMENTAL VALIDATION WORK

As detailed in sub-section 3.3, validating the 1D/3D digital coupling model requires a good degree of control over the various components of the research process. This is because to validate 1D/3D coupling, there is a need to know how to determine the effect of errors relating to the 1D or the 3D part of the calculation, or, and this is just as important, relating to the experimental validation data.

5.1 DIGITAL VALIDATION

The main part of basic New-Vendis development work was completed in 2000 with the programming of the basic effects of a network: momentum load losses and fan operating trends. Allowance for variations in air density on these basic effects was also programmed. An important validation step was achieved using this basic version of the 1D code. This allowed validating the different parts of the code (grid determination, scenario declaration, resolution, result post-processing) on simple cases and evaluating the tool's possibilities when applied to complex cases [10]. This validation step then allowed the setting up and testing of more complex effects such as mobile effects and the setting up of precise heat transfer models. Then, lastly the communication system between New-Vendis and Phoenics was installed. This document only presents some of the validation steps from among those actually carried out at various stages during development.

5.1.1 Case of a Network with a Fan

The following test case also served to validate the preparatory part prior to calculation, but we will not be detailing the results obtained here. We tested three effects:

- Air flow induced by a branch with a set air flow
- Air flow induced by a fan
- Determining the characteristic point of an auxiliary fan

The reference item for this validation work is Scott and al. [15].

The network configuration was as follows:
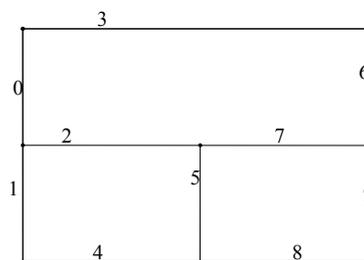
Figure 5—1: Illustration of the branches in the reference network.

Here only cases of fan induced air flow are presented.

The fan trend is shown by the following points:

| Flow | 0 | 0.5 | 1 | 1.5 | 2.5 | 2.75 | 2.77 | 2.78 | 2.95 | 100 |

| Pressure | 35.60 | 33.60 | 32.80 | 31.20 | 28.40 | 26.50 | 26.30 | 26.20 | 24.40 | 28.80 |

Table 1: Fan characteristics

Branch	0	1	2	3	4	5	6	7	8 and 9
Reference flows	-1.167	0.833	0.333	-1.167	0.833	1.667	-1.167	-1.333	2.5
Calculated flows	-1.168	0.834	0.334	−1.168	0.834	1.666	−1.168	−1.332	2.5

Table 2: Results obtained

The stop test value corresponds to the minimum value that the sum of contributions (in the grid during a complete cycle) needs to reach so that calculation can be considered to have converged. In this case, this value is worth 10^{-3}, the calculated results are clearly precise to within 10^{-3}.

5.1.2 Validating the Heat Transfer by Convection and Radiation

The heat transfer validation was performed by reproducing the theoretical model in both the spreadsheet and in New-Vendis. The independence of the time increment chosen in New-Vendis compared with the stationary solution actually obtained was checked. This example uses local 1D refinement.

The results obtained which use both the heat transfer by convection and by radiation show (see figure below) the very good match obtained between the theoretical results and the simulation results.
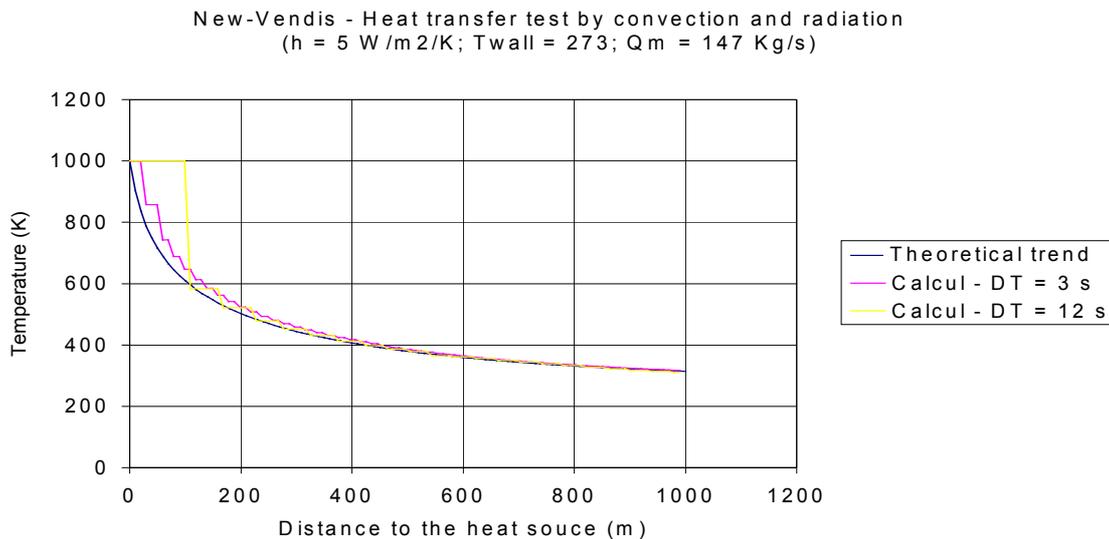
Figure 5—2: Example of fluid/wall heat transfer validation – comparison between the theoretical result and the result produced by New-Vendis.

Note that working effects do exist, they are linked to the general fluid propagation model used in the branches. However, the theoretical solution is well complied with by the model. This solution is improved when the local refinement level (DT=3s) is increased, which is the effect sought. Note also that the remains close to the solution while overestimating it by steps when the refinement level degraded (DT=12s) something which appears to err on the safe side.

5.1.3 Validating the Non-stationary Aspect

This effect is modeled by estimating a back pressure which translates the inertia effects of fluid masses. In terms of inertia, we estimate this back pressure by applying the fundamental principle of dynamics to the fluid mass (air in this case) contained in the affected branch and we state that this force equals the back pressure over the branch section. The following expression is obtained for P_{instat}:

$$P_{instat} = \rho L \frac{Q_i^{t+1} - Q_i^t}{S_i \Delta t}$$

The reference article [16] contains actual measurements taken in tunnels. The tunnel with a length of 6,560 meters and a fall of 57 meters is covered by a draft of initial natural air at a speed of 1 m/s. There is an air mass of some 400 tons in the tunnel. We assume that this is dry air. The tunnel wall roughness factor is 0.025. The graph below shows a comparison between the rising and falling air velocities within the tunnel when the fans are started and stopped.
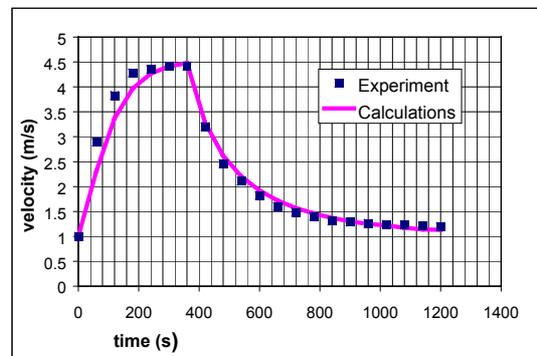
Figure 5—3: Change over time of air flow velocity within the tunnel.

The results obtained by New-Vendis are highly satisfactory. We can see that the start up phase is a little less well simulated than the return to the initial flow velocity. We explain this by the fact that the assumptions made for this test case are slightly imprecise, especially in terms of density which was estimated and retained as constant.

5.2   EXPERIMENTAL VALIDATIONS

This heading refers to 'experimental validations' as the validation of measurements actually taken from the test set up.

As stated above, it is indeed important to know the experimental errors so as to know what acceptable differences there will be between the numerical results and the experimental results when the two are compared.

As a general rule, for all of the measurements made, special attention has been paid to their quality. Furthermore, the tests were repeated a number of time in order to ensure that test conditions are fully under control. This is extremely important when it comes to validating numerical developments for which the initial and limit conditions must be well known.

Some indications are provided below on the checks made on the readings taken during the tests as well as on the data acquisition mechanism.

5.2.1 Pressure Sensors

The measurement precision is +/-0.25% of the reading. For each device, a calibration certificate connected to COFRAC is available. Preliminary tests made it possible to define the operating ranges of the devices based on their location in the test set up. Consequently, pressure transmitters in the +/-50 Pa to +/-3000 Pa range were placed at the ten measurement points.

5.2.2 Air Velocity Sensors

5.2.2.1 Windmill Anemometers

The windmill rotation speed varies practically linearly with the air velocity, furthermore it is practically independent of parameters such as pressure, temperature, density and humidity. The measurement range is 0.5-20 m/s. The measurement precision is +/-3% of the reading. For each device, a calibration certificate connected to COFRAC is available.

5.2.2.2 Hot Wire Anemometers

The gas velocity is proportional to the drop in temperature on the hot wire. This measurement principle is therefore sensitive to the make up of the gas, given the calorific capacities of gasses. The measurement range is 0.1-20 m/s. Measurement precision is +/-0.03% of the display and +/-2 digits for the transmitter and +/-0.06 m/s for the sensor. Sensor precision comes from a calibration certificate connected to COFRAC. This sensor has be located at a position where the gas that circulates is only air (calibration being performed under the same conditions).

5.2.3 Concentration Sensors

The helium content is monitored by a mass spectrometer set for the mass in helium. The device is designed for a linear measurement range of between 10^{-6} and 1% in volume. The helium content levels to be measured being between 0 and 100% (0%=ventilation air, 100%=pure helium representing fire smoke at a temperature of around 2000K). A system for diluting the gasses to be analyzed has been implemented in order to place it within the range (0-1%). The spectrometer is calibrated at a value of 1% using mass flow meters. A study comprising generating helium with different content levels and analyzing the helium (spectrometer) and oxygen (paramagnetic SERVOMEX analyzer) in parallel, showed the linearity of the spectrometer and its ability to successively measure low and high helium content levels. The relative precision of the measurement was estimated at +/-5%.

5.2.4 Data Acquisition System

The data acquisition system comprises PC type portable computer running graphic language programming software (LabView 6i) and data acquisition equipment by National Instruments.

The data acquisition equipment comprises a measurement system and a PCMCIA input/output card.

The I/O card is a PC-AO-2DC type card. It therefore has two voltage outputs and two analog current outputs. A CB-27 type connector block handles the link between the voltage outputs (0-10 V) on the card and analog fan control inputs. The analog/digital converter (ADC) on the card has 12 bit resolution. Its precision is +/- (0.0005 V + 1.5.10^{-3} U) where U represents the output voltage.

The measurement system comprises SXCI type hardware. SCXI (Signal Conditioning eXtensions for Instrumentation) is an instrumentation and signal conditioning system suited to data acquisition on PCs:

Parallel port
Conditioned signals
SCXI rack
SCXI module
Connector block
Sensors and I/O signals
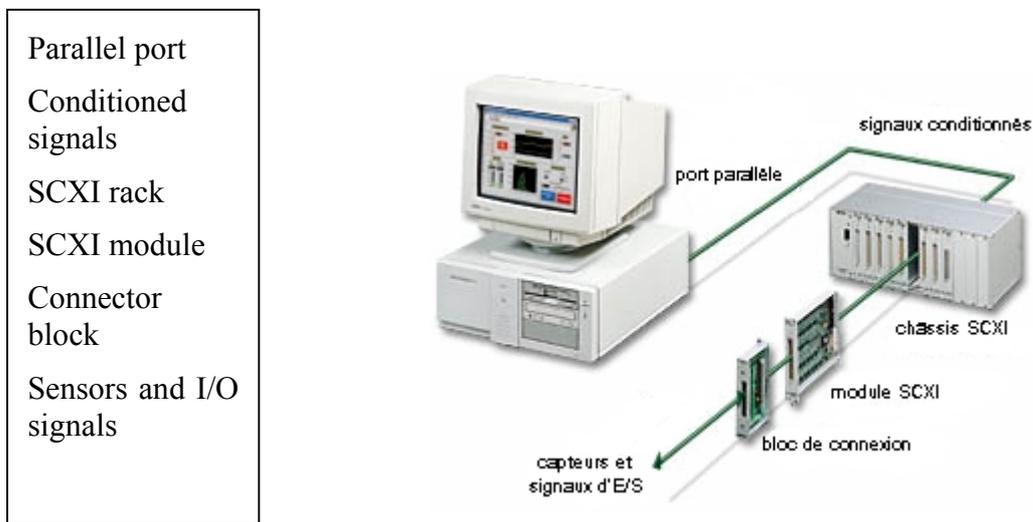
Photo 5—1: Data acquisition system.

The modules were calibrated and their measurement deviations (brought back to two standard deviations) are as follows:

     - SCXI-1200 + SCXI-1302: +/-(0.001 V +1.3.10^{-3}.U),

     - SCXI-1100 + SCXI-1300: +/-(0.0021 V + 3.10^{-3}.U),

     - SCXI-1100 + SCXI-1308: +/-(0.008 mA + 3.2.10^{-3}.I).

The measurement range covered by the SXCI-1200 module's ADC is 0-10 V, with 12 bit resolution. The ADC's resolution is therefore q = 10/(2^{12}-1) = 2.44 mV for voltage measurements mm measurements and 0.001 mA for current measurements.

Data acquisition and register and fan control is performed from a VI (Virtual Instrument) developed in LabView. This VI handles configuring the card and the SCXI modules (gain, measurement channel selection, sampling frequency,…) and checking acquisition. A Measurement & Automation Explorer (MAX) interface is used to enter data for each measurement channel (channel name, description, measurement physical units, scale, calibration curve,…). The data in physical units is recorded in real-time in a file on the computer's hard disk drive.

6. CONCLUSION AND PROSPECTS

This summary report provides an overview of the work undertaken during this project.

In actual fact, readers will have understood that this project lays the groundwork for a new development for research into the field of safety in confined spaces, whether underground or not. The contribution of this project can be illustrated by identifying three sub-projects which could either reach completion at an operational engineering level in the short or medium term, or serve as the basis for developing new research work.

The first one covers the development of a new code for calculating ventilation networks, taking into account accident scenarios: New-Vendis. This calculation code now benefits from current computer system progress, including that relating to Internet communication networks.

The second one covers taking into account communication networks in New-Vendis, so as to simulate an overall flow, bringing together the respective strong points of different kinds of calculation codes for fluid engineering. This is what has been called code coupling. In future, these simulation principles may become an efficient way to calculate complex accident scenarios. This project has produced the first computer bases for calculation code coupling as well as the concepts destined for future developments.

The third one covers the setting up of a new test laboratory for the small-scale simulation of accident situations in confined spaces. During this project, the design of this laboratory and its instrumentation, qualifying the scheduled tests and their instrumentation, the study and feasibility of other kinds of tests in confined locations and the production of validation data for digital code coupling were all achieved.

Each of these three sub-projects has reached a different stage.

The first and third ones are operational as research tools into safety in confined environments and have in this project produced their first results: simulating in-tunnel accident situations and producing experimental data for validation purposes.

The second one is lest advanced but some of the planned concepts have already been validated: New-Vendis code structure operational for code coupling, computer communication principles between codes implemented and validated.

Lastly, this project represents a starting point from at least two points of view. On the one hand, the New-Vendis research tool was thought out to become adaptable in the medium term to different risk engineering in confined locations contexts (road or rail transport, mining, storage, mains networks…). New-Vendis could therefore represent a starting point for producing tools that are specifically suited to a given context. Furthermore, the experimental developments undertaken during the project, and especially the setting up of a new test laboratory open an additional way forward in studying accident hazards in confined spaces: model simulation. This way forward complements those that already exist at INERIS, including: full scale testing and digital simulation.

7. BIBLIOGRAPHY

[1]   **International Congress CNPP-INERIS** "Fire Safety in Hazardous Enclosed Spaces-Tunnels, Underground Spaces, Parkings, Storages", Vernon, France, November 1999.

[2]   **Lacroix, D.,** "Capabilities and Complementarities of Full Size Tests, Model Experiments and Numerical Simulations, for the Study of Tunnel Fires.", International Conference on Fire Protection in Traffic Tunnels, Dresden, Sept. 12-13, 1995.

[3]   **Rhodes, N**., "CFD Code Predictions of Memorial Tunnel Fires Tests", Interflam'96.

[4]   **E. Ruffin, A. Carrau, C. Cwiklinski**, Numerical Simulations of Locomotive Fires in the Lyon-Turin Tunnel, ITC Conference on "Tunnel Fires and Escape from Tunnels", May 5-7, 1999, Lyon, France.

[5]   **Charters, A., C., Gray, W.A., McIntosh, A., C.,** "A computer Model to Assess Fire Hazards in Tunnels (FASIT), Fire Technology, First Quarter 1994, pp 134-154.

[6]   **Gunther J., d'Albran N.** "VENDIS, Ventilation Network with Color Display", CdF/CERCHAR, 1984.

[7]   "Emergency Response to Tunnel Fires". Working group led by the French 'Department of Defense and Civil Security'. January 2000.

[8]   **Cross, H.,** "Analysis of Flow in Networks of Conduits or Conductors", Bull. Ill. Engng. Exp. Sta No. 286,Urbana, 1936, 29p.

[9]   **E. Ruffin, S. Daëron**, "Perfecting a Simulation Tool for Accidental Situations in Underground Networks", annual progress report 1999, INERIS.

[10]   **S. Daëron, E. Ruffin** "Calculations of Fire Smoke Behavior in Long Rail Tunnels", First International Conference on Tunnels and Underground Station Fires, Hong Kong, May 3-4, 2000.

[11]   **T. Maëda, M. Kinoshita, H. Kajiyama and T. Tanemoto, "**Estimation of Aerodynamic Drag of Shinkansen Trains from Pressure Rise in Tunnel", proceedings of sixth international symposium on the aerodynamics and ventilation of vehicle tunnels, Durham, England, 1988.

[12]   **G. Marlair, JP Bertrand and S. Brohez**. "The Use of the ASTM E 2058 for the Evaluation of Under-ventilated Fires". 7th Int. Congress of Fire and Materials, San Francisco, CA, Jan. 2001 (22-24) , Proc p 301-313.

[13]   **Ruffin E., Schiestel R., Anselmet F., Amielh M., Fulachier L**. "Investigation of Characteristic Scales in Variable Density Turbulent Jets Using a Second-order Model". Physics of Fluids A, vol. 8, No. 6, August 1994.

[14]   **D.R., Scott, F.B., Hinsley**, Ventilation Network Theory, Colliery Engineering, 1951, 1952.

[15]   **Lotsberg, G.**, "Measurement of the Wall Friction Factor and the Installation Efficiency of Jet Fans", 9th International Symposium on Aerodynamics and Ventilation of Vehicle Tunnels, Aosta Valley, 1997.