LIEGE UNIVERSITY
Urban & Environmental Engineering
Structural Engineering

USER'S MANUAL FOR SAFIR (version 2022)

A COMPUTER PROGRAM FOR ANALYSIS OF STRUCTURES

SUBJECTED TO FIRE

Part 1: general considerations

by

Jean-Marc Franssen* & Thomas Gernay**

07/05/2023

* jm.franssen@uliege.be – Liege University, Liege, Belgium
** tgernay@jhu.edu – Johns Hopkins University, Baltimore, MD, U.S.A.

Table of content

1.   Notations

Files:

"*This is a fixed file name*"

[This is a variable file name], (depending on the user)

"This is a type of file", (for example : ".IN"; ".txt")

Parameters and command

```
THIS IS A PARAMETER OR A COMMAND
```

This is replaced by the value assigned to the parameter or following the command

```
[This is a variable command],
```
(depending on the user), (for example :
```
[filename]
```
)

Example :

- NNODE is a command that indicates to SAFIR the meaning of the numbers that are present on the same line or in the subsequent lines after the command.
- *nnode* is the quantity of nodes, following the command NNODE

Examples and notes:

```
This is an example
```

This is a note

2. Description of SAFIR

2.1 Description of the model

SAFIR is a special purpose computer program for the analysis of structures under ambient and elevated temperature conditions. The program, which is based on the Finite Element Method (FEM), can be used to study the behaviour of two and three-dimensional structures. SAFIR was originally developed by Professor J-M Franssen at the University of Liège, Belgium, but the development is currently also ensured by Professor T. Gernay at Johns Hopkins University, USA.

The underlying model for computing temperatures in the structure is that in which heat is distributed in the structure essentially by conduction because most of construction elements are made of solid. Radiation and convection are the heat transfer modes in internal cavities such as those present, for example, in hollow core slabs[1]. Radiation and convection are the heat transfer modes at the boundaries between the object analysed and the environment, i.e., the fire.

The underlying model for computing the mechanical behaviour of the structure is based on oriented elements of structural mechanics such as truss, beam, and shell finite elements. Some models can also be built with 3D solid finite elements that follow mechanics of solids. Constitutive models are based on the hypothesis of strain decomposition in the frame of associated plasticity for most material, and a combination of damage and plasticity theory for the concrete models that include explicit transient creep[2].

2.2 Description of the basic processing tasks, calculation methods and procedures

2.2.1 General layout of a calculation

Using the program, the analysis of a structure exposed to fire consists of several steps. The first step involves predicting the temperature distribution inside the structural members, referred to as *«thermal analysis»*. The second step of the analysis, termed the *«structural analysis»*, is carried out for the main purpose of determining the response of the structure due to static and thermal loading. The «*torsional analysis»* may be necessary for structural analyses using 3D BEAM elements. The various stages of analyses are briefly outlined in the following sections.

---

[1] Internal cavities can be considered only on 2D thermal calculations.
[2] ETC in the name of the material?

2.2.1 Input for SAFIR

For any analysis using SAFIR, data files acting as input files to the program are prepared. For each analysis type (thermal, torsional or structural analysis), the user prepares one data file. This is an ASCII file, created with a text editor, in a word processor, by users' specific script or by the pre-processor GmSAFIR, and it must have the filetype ".IN".

This file with a ".IN" extension contains information such as calculation strategy, time discretization, loads, node coordinates, types of finite elements used, material properties, etc. For structural analysis, the ".IN" file specifies the name of the files created during thermal analyses and in which the temperature data is stored.

Figure 1shows a schematic representation of the different steps and files that may be involved in the case of a 2D frame structure comprised of two types of different sections, one for the columns and one for the beam. The structure is modelled by BEAM finite elements. The user must create the ".IN" files. The commands, format and quantity of lines required for a section in the input files are detailed in Parts 2, 3 and 4 of the manual.
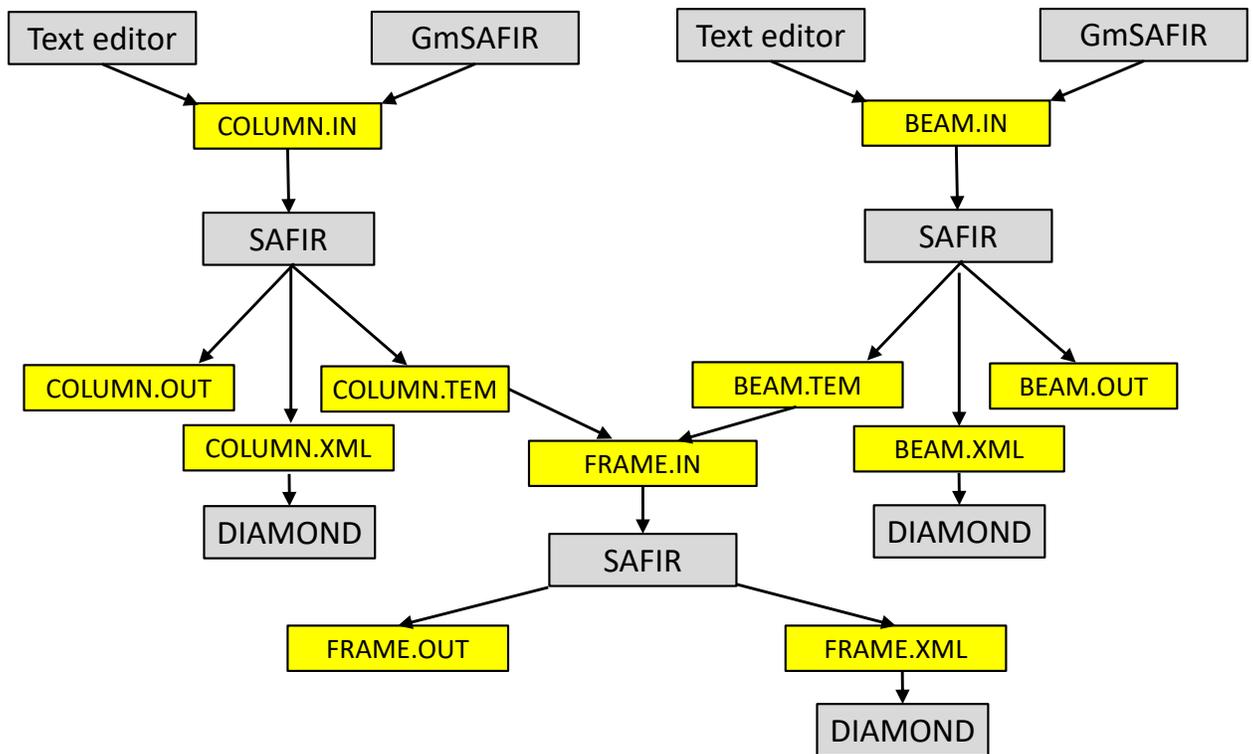
Figure 1: flowchart of the analysis of a 2D portal frame

2.2.2 Reading the input files

SAFIR first reads the whole input file and copies it in the output file.

Then it reads all lines of the input file again, one by one, tries to interpret the data that are found in each line and prints the interpreted results in the output file before going to the next line.

If the command that is found in the input file is not the one that was expected at this stage, an error message is written in the ".OUT" file and the run is terminated.

Some checks are made on the plausible character of the input numbers. If, for example, a negative value ha been entered as a material strength, an error message is written in the ".OUT" file and the run is terminated. It must yet be considered that not all variables are checked with every possible mistake that can be made by the users. A mistake that has not been checked by SAFIR whereas this would have saved time to a user should be reported to the developers who will then introduce this check in the next version of SAFIR.

If the format of the input data does not correspond to the one that is expected, for example if a chain of character is found when a real number was expected, SAFIR cannot recover this error and a message of the kind "lforrtl: severe (59): list-directed I/O syntax error, unit…" will be printed in the DOS window or in the run window of SafirShell in which SAFIR is run.

It is thus a good idea, if a run terminates prematurely, to open the ".OUT" file and either read the error message that may have been written by SAFIR or, if not, to see the amount off data that could be read and interpreted correctly before the run terminates. This can give a clue about the location in the input file where the error should be found.

The checks mentioned here deal with the grammar of the input file. Conceptual errors such as a load being applied that exceeds the load bearing capacity of the structure cannot be detected by the code.

2.2.2 Computing temperatures in the structure

This analysis is usually performed while the structure is exposed to fire. For a complex structure, the sub-structuring technique is used, where the total structure is divided into several substructures and a temperature calculation is performed successively for each of the substructures. This kind of situation does arise in a structure where the members are made of different section types or made of sections submitted to different fire exposures. The thermal analysis can be made using 2D SOLID elements, with the results to be used later in the mechanical analysis on cross sections of BEAM elements or on the thickness of SHELL elements. If the thermal analysis is made using 3D SOLID elements, the results will be used in a subsequent mechanical analysis also using 3D SOLID elements.

A.   Temperatures in beams

The temperature is non-uniform in the sections of beams, but there is no heat transfer along the axis of the beams. As an example, a frame structure with reinforced concrete columns, pre-stressed main beams, and structural steel secondary beams, will require a separate temperature analysis for each of these three section types. From these analyses, the temperatures across the cross section are obtained and are stored for subsequent structural analyses where these sections are present.

B. Temperatures in shells

The temperature is non uniform on the thickness of the shell, but there is no heat transfer in the plane of the shell. The temperature analysis is performed on a section having the thickness of the shell (and its eventual thermal protection) and an arbitrary width, 1 cm for example. The node numbering is from 1 to (*nnode* / 2) from the bottom to the top of the section and, again, from (*nnode* / 2 + 1) to *nnode* for the second row of nodes, see Figure 2.

Figure 2: model for the temperatures across the thickness of a shell

A file named [xxxx.TSH] is created in which the temperatures of the first (*nnode* / 2) nodes are written. Below is given an example of such a file. For particular cases, like for example a uniform temperature distribution, a similar file can be created with a text editor. The quantity of elements and the size (here the thickness) of these elements is independent from the quantity and position of the points of integration that will be used later in the structural analysis. In the structural analysis, the temperatures at the points of integration will be linearly interpolated from the temperatures of the nodes.

```
THIS IS A COMMENT LINE


       HOT
POSITIONS OF THE NODES.
========================
NUMBER OF POSITIONS:  11
-0.5000E-01 -0.4000E-01 -0.3000E-01 -0.2000E-01 -0.1000E-01 0.0000E+00
0.1000E-01 0.2000E-01 0.3000E-01 0.4000E-01 0.5000E-01


TIME = 60.0000 SECONDS   OR   1 MIN.  0 SEC.
==========================================
   -0.0500      56.41
   -0.0400      25.16
   -0.0300      20.15
   -0.0200      19.96
   -0.0100      20.00
    0.0000      20.00
    0.0100      20.00
    0.0200      20.00
    0.0300      20.00
    0.0400      20.00
    0.0500      20.00
```

SAFIR will calculate the temperature in the structure at successive time steps based on the choices of the user. For each time step that starts at time t_n:

- SAFIR first extrapolates the temperatures from the two previous time steps, t_{n-1}, and t_n to a first approximation at a given time $t_n + Dt$[3] comprised between t_n and t_{n+1}. The temperatures at this stage are not in equilibrium.
- SAFIR refines this first approximation of the temperatures at $t_n + Dt$ until thermal equilibrium is restored. Several iterations are required to restore the equilibrium up to an accepted degree of precision[4].
- SAFIR finally extrapolates the temperatures at t_{n+1} from the values at t_n and $t_n + Dt$.
- If the time step is one of the time steps defined for printing the results, see `TIMEPRINT` command, the temperatures at the nodes are printed in the output file. Temperatures are also eventually written in the ".TEM" or ".TSH" file to be used in a subsequent analysis, and in the ".XML" file to be read by the graphic postprocessor DIAMOND.

---

[3] The value of Dt is governed in the data by the parameter `TETA` = Dt / (t_{n+1} - t_n)

[4] The degree of precision required before thermal equilibrium is considered as acceptable is governed in the data by the parameter `PRECISION`.

These processes are shown schematically in Figure 3 and Figure 4.
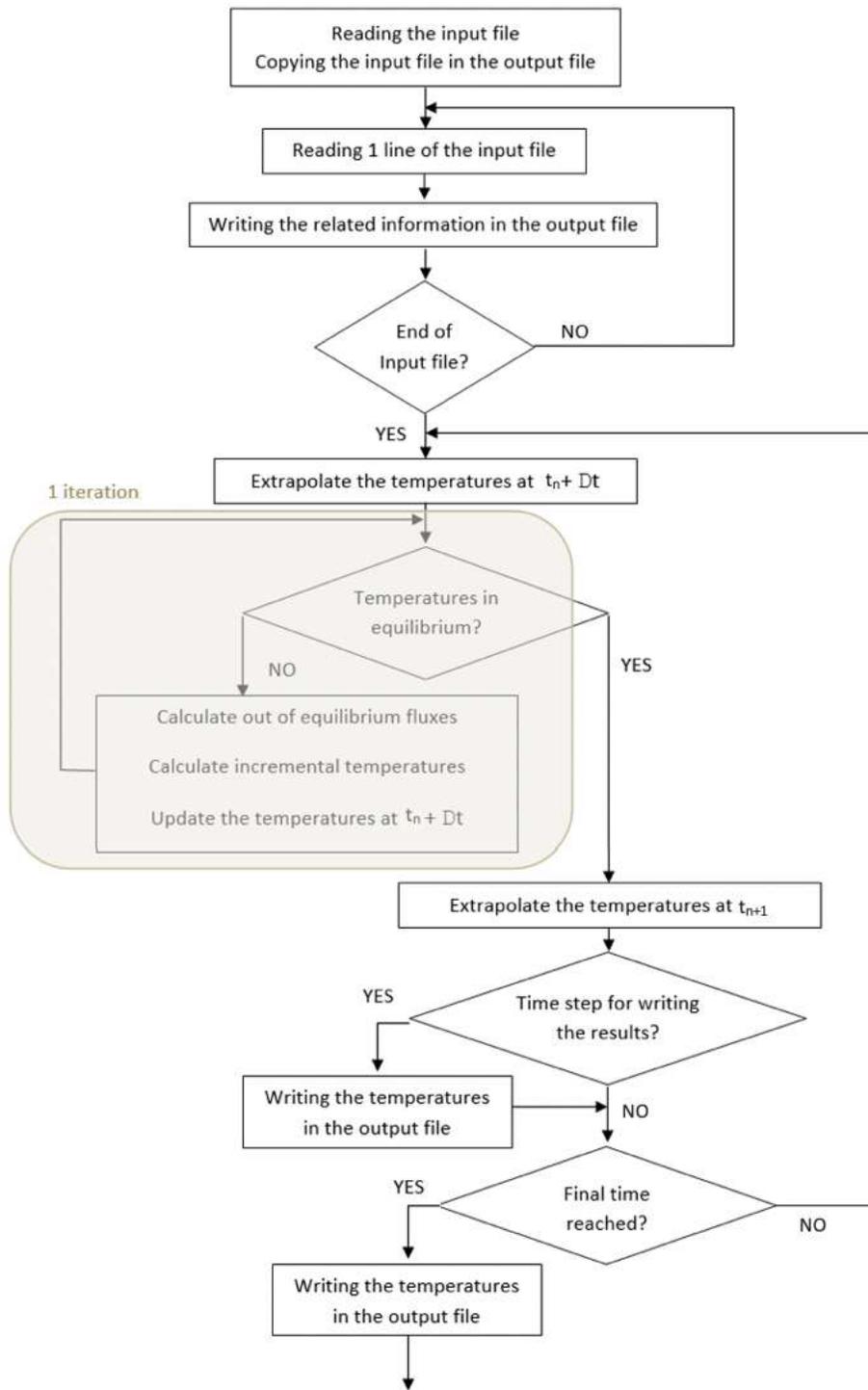
Figure 3: graphic representation of the algorithm for a thermal calculation
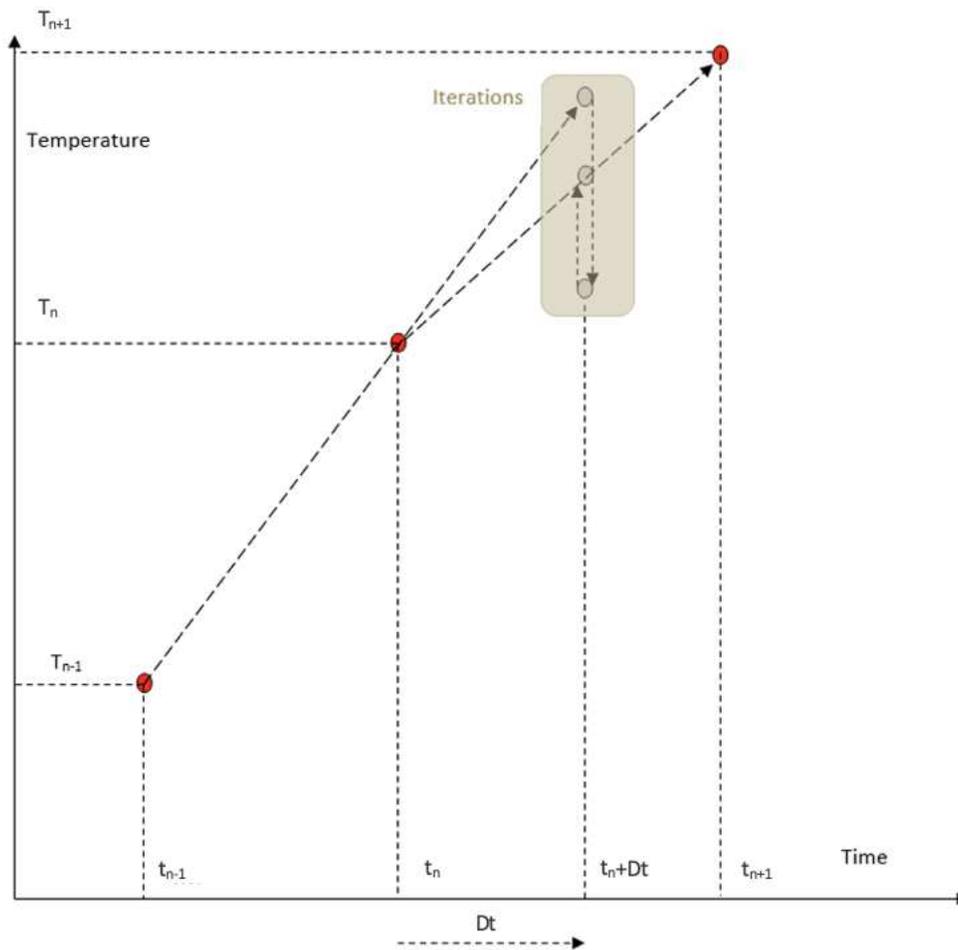
Figure 4: graphic representation of the iterations performed within one time step

2.2.3 Computing the mechanical behaviour of the structure

For each calculation, the loads are applied to the structure. The temperature history of the structure, due to fire, is read from the files created during the thermal analysis. As the computation strategy is based on a step-by-step procedure, the displacements at each node of the structure are automatically given at selected time steps. The following information can also be printed if this has been requested in the input file. This information cannot be retrieved once the calculation is performed if it has not been requested:

- Axial and shear forces and bending moments at integration points in each finite element.
- Reaction forces or moments at the supports.
- Strains, stresses and tangent modulus at integration points of each finite element.

The transient mechanical behaviour of the structure is also computed in a series of subsequent time steps, with equilibrium being approached in each time step by a series of iterations.

At every point of integration in the structure:

- At a given time step t_n with corresponding temperatures T_n, the stress-related strain is $\varepsilon_\sigma(T_n)$ and the plastic strain ε_{pl}. The situation is represented by point A on Figure 5.
- The temperature computed in the thermal analysis at time T_{n+1} is read from the file where it has been stored during the thermal analysis.
- The increment of thermal strain $\Delta\varepsilon_{th}$ at every point of integration is calculated based of temperature increment from *n* to *n+1*.
- The properties of the material are updated corresponding to T_{n+1}. As a result, the virgin stress-strain law (σ - ε_σ) is different at temperature T_{n+1} compared with T_n.
- The displacements are kept constant at the beginning of the time step, the total strain ε_{tot} is thus also constant. So the $\Delta\varepsilon_{th}$ generates a new value of stress-related strain, $\varepsilon_\sigma(T_{n+1})$. For a temperature increase, the structure is "artificially compressed".
- Taking into account the new value of the stress related strain $\varepsilon_\sigma(T_{n+1})$, the stress-strain law at T_{n+1}, and the fact that the plastic strain is constant (ε_{pl}), the new stress and tangent modulus are calculated (point B on Figure 5).
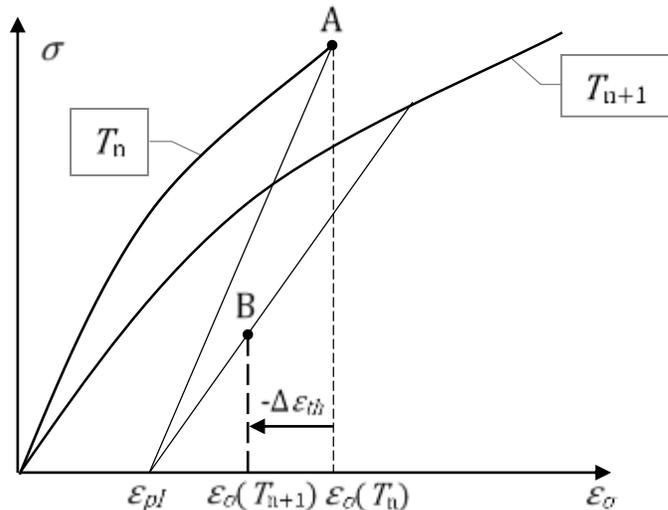

Figure 5: evolution during the first iteration

The stresses are integrated on the volume of the elements to compute the internal nodal forces which are not anymore in equilibrium with the applied nodal forces. In case of temperature increase, the structure undergoes an internal state of out of balance compression.

The stiffness at the integration points is integrated on the volume to compute the stiffness matrix of the structure.

The out of balance forces are applied to the structure, leading via the new stiffness matrix to incremental displacements, new strains (stress related component), new stresses and new nodal forces.

The procedure is repeated several times at constant temperature until the convergence criteria is satisfied. Each of these repetitions is called an iteration.

Plastic strains are updated after convergence which completes the time step.

2.2.4 Analysis of torsional stiffness of BEAM elements

This analysis is performed when analysing structures with 3D BEAM elements, because non-uniform torsion is implemented in the BEAM finite element and cross-section are thus subject to warping and because the torsional stiffness of section is not always available from tables or formulas. The 2D SOLID elements are used to calculate the warping function and the torsional stiffness of the cross section.

The torsion analysis must be performed on the same section and with the same discretisation as those used for the thermal analysis. If the input file for the thermal analysis was named [name.in], then the input file for the torsional analysis in the same section must be named [name-t.in].

If the structural model is using 2D BEAM F.E., only the file [name.tem] will be used. If the structural model is using 3D BEAM F.E., the same file [name.tem] will be used, but also the file [name-t.tor] produced during the torsion analysis.

2.3   Capabilities of SAFIR

SAFIR can be used for performing three different types of calculations, namely, thermal, torsional and structural analysis. The capabilities of the program concerning these three analysis types are outlined in this section.

2.3.4 Capabilities concerning the thermal analysis

- Plane sections as well as three-dimensional structures can be analysed.
- Plane sections are discretized by triangular and/or quadrilateral (rectangular and non-rectangular) elements, allowing representation of virtually all cross-sectional shapes.

- Three-dimensional structures are discretized by SOLID elements (prismatic and non-prismatic) with 4, 6 or 8 nodes. This allows the representation of virtually all structure shapes.
- Variation of material from element to element is possible.
- The fire temperature, defined as a function of time, can either be the standard curves predefined in the code (ISO 834, ASTM E119, HCM or RWS) or any other curve introduced through data points.
- Cooling down phases can be considered, as long as the material thermal properties in the cooling phase are known.
- Variation of material properties with temperatures, as well as the evaporation of moisture, can be considered.
- SAFIR can analyse models made of materials such as steel, concrete, wood and aluminum, the properties of which are embedded in the code. Other materials can also be defined by the user.
- Automatic adaptation of the time steps is possible.

2.3.5 Capabilities concerning the structural analysis

- 2D or 3D structures can be analysed.
- The structure is discretized by means of five different element types: TRUSS elements, made of one single material with one uniform temperature per element; BEAM elements, SHELL elements; SOLID elements and SOIL elements.
- Large displacements are considered in the TRUSS, BEAM, SHELL and SOIL elements.
- The effects of thermal strains (thermal restraint) can be accounted for.
- Material properties are non-linearly temperature dependent.
- Unloading of material is typically parallel to the elastic-loading branch (plasticity), except for concrete in which a plastic-damage model has been considered.
- In situ concrete, that sets after some loads have been applied, can be considered.
- Local failure of a structural member that does not endanger the safety of the whole structure can be handled by means of a dynamic analysis.
- Imposed displacement (prescribed degrees of freedom) can be introduced.
- External support inclined at an angle to the global axes can be considered.
- Residual stresses (initial strains) can be accounted for.
- Pre-stressed structures can be analysed.
- Automatic adaptation of the time steps is possible and structural calculation continues until failure or, alternatively, when the maximum deflection has reached a value defined by the user.

- Time-dependent, elasto-plastic, semi-rigid connections can be applied at the end of beam finite elements.

2.3.6 Capabilities concerning the torsional analysis

- Utilisation of triangular and quadrangular elements allows virtually all cross-section shapes to be represented.
- Materials are in the elastic stage, at constant temperature (and, hence, the torsional stiffness remains constant during the simulation of the structural behaviour).
- If the user wants to adjust the torsional stiffness in order to take into account an increase of temperature during the fire, this is done by giving a reduced value of the Young's modulus in the material properties used for the torsional analysis.

2.4 <u>Common features in all analyses</u>

The common features in all computations are:
- The system of equation is solved by the sparse matrix solver PARDISO.
- The same temperature or the same displacement can be imposed at two or several nodes using master-slave relationships.
- Thermal and mechanical properties of steel, concrete, wood and aluminium according to the relevants Eurocodes are embedded in the code and can be used directly.
- Graphic pre-processing is made using the code GmSAFIR and post-processing using the code DIAMOND. Both codes are available for free, GmSAFIR on the GmSAFIR wiki (https://github.com/gmsafir/gmsafir) and Diamond on the SAFIR web site (https://www.uee.uliege.be/cms/c_6331644/fr/uee-safir).

2.5 <u>Description of the types of skills required to execute typical runs</u>

The user should be able to operate a P.C. computer, either a laptop or a desktop.

The user should be familiar with the Windows operating system.

The user should be familiar with the file naming convention based on a file name and a file type. He should be able to use the Explorer of Windows to move from one folder to another, to copy, delete, or rename files. It is highly recommended to set Windows Explorer in such a way

that the filetype of all files are shown. Experience has shown that hiding the filetype of the common files may lead to significant waste of time. One example is when the Explorer shows a file as "filename.txt" whereas the real name is in fact "filename.txt.txt" and the user does not realise that, if the full name was "filename.txt", it should be displayed as "filename".

The user should be able to open files with a text editor, to prepare or modify input files and to examine the results produced with SAFIR.

To execute typical thermal analyses, the user should be familiar with the basic mechanisms of heat transfer such as conduction, convection, and radiation. He should also be familiar with the finite element technique applied to solving heat transfer problems.

TO execute typical structural analyses, the user should be familiar with the theories of structural mechanics, including the behaviour of beams and shells and the effects of large displacements on local, member or global instabilities.

To judge from the relevant character of the results the user should be familiar with the behaviour of structures subjected to fire. For example, a user that has all skills and experience mentioned above would be able to perform a numerical simulation of a section protected by a gypsum plaster board and subjected to a fire for 4 hours, only missing the fact that no gypsum plaster board can stay in place for such a long duration (it would disintegrate long before that).

3. Installations and operating instructions

3.3　Minimum hardware configuration required

SAFIR can run on a P.C with a single processor. 1.6 GHz.
1 Go of RAM
120 Go Hard Disk
This has been tested on an ACER Aspire One ZG5 notebook.

SAFIR could **not** run on a Pentium II, 350 MHz, 384 Mo RAM, 37,2 Go ROM.

3.4 Computers on which SAFIR has been executed successfully

- Laptop HP Pavilion Entertainment PC
  Intel Corel2 CPU, 1.66 GHz, 2 Go RAM, 232 Go Hard Disk

- Dell Precision T5400
  Intel Xeon CPU E5420, 2.50 GHz,8 Go RAM, 300 Go Hard Disk

- Asus under Windows XP Professional Version 2002, Service Pack 3
  Intel Pentium 4, 2.60 GHz, 1 Go RAM, 29 Go Hard Disk

- ACER Aspire One ZG5
  Intel Atom 1.6 GHz, 1 Go RAM, 120 Go Hard Disk

3.5 Programming language, software operating system and version in use

SAFIR has been written in FORTRAN:

Intel Visual Fortran Compiler (Package ID: w_comp_lib_2020.1.216) under Microsoft Visual Studio 2019, version 16.10.4.

Some routines, written at the early days of development of the code, are in FORTRAN 77 while the most recent subroutines are in FORTRAN 90.

3.6 Instructions for installing the program

SAFIR is delivered in the form of an executable file named *"safir.exe"*, and a security file named *"identity.key".*

Installation of SAFIR simply requires copying these two files on the memory devices connected to the computer(s) where SAFIR will be run.

The file "*safir.exe*" must be copied on the device where it will be found by the C.P.U of the computer during execution. This could be, for example, a USB key, an external hard disk or, more likely, one of the hard disks of the P.C.

The location of the file "identity.key" depends on the method which will be used to run SAFIR, see section 3.7

3.7 RUNNING SAFIR

3.7.1 From the Windows Explorer

The file *"safir.exe"* can be in the folder of choice of the user.

With this method, the file *"identity.key"* must be located **in the root directory** of the device where "safir.exe" has been located, for example in c:

Users normally organize their input files in different folders; one folder may for example be used for each different project. The executable can be copied in each of these folders, see Figure 6, in which "*safir.exe*" has been renamed into "*safir2000.exe*".
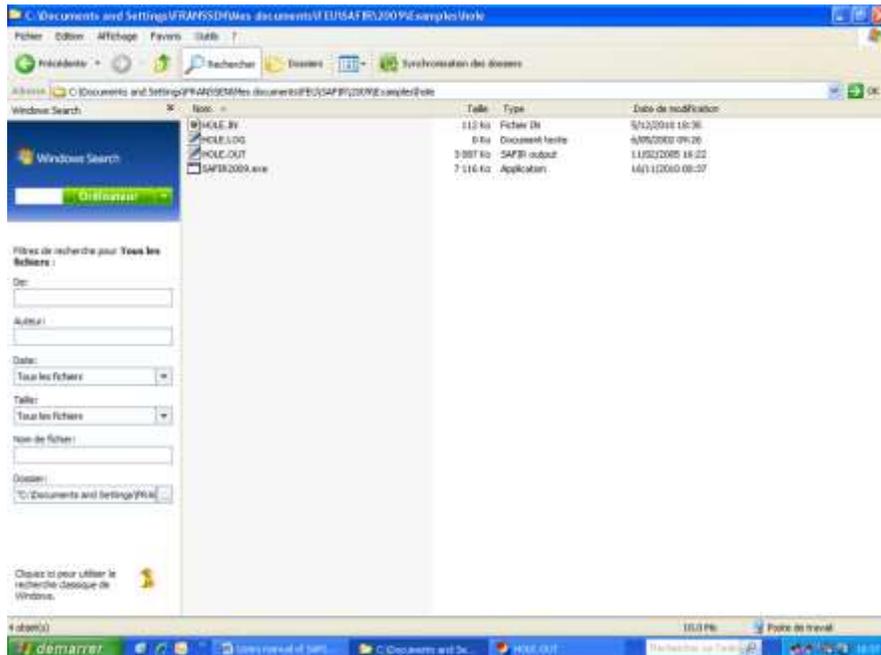

Figure 6 folder with the executable

To run SAFIR from Windows Explorer, double click on the executable file. SAFIR will open a DOS window in which the user is requested to type the file name of the input file ([`filename`] from the complete name [filename.IN]), see Figure 7.
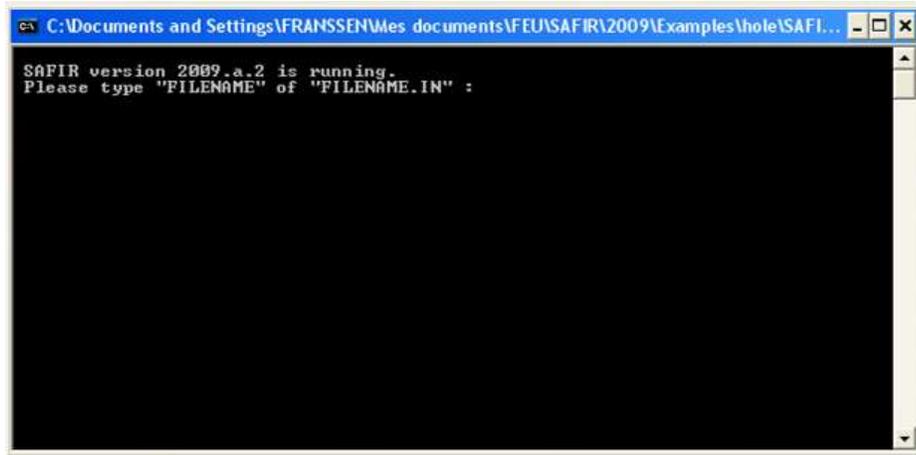
Figure 7: DOS window with the request for the file name

The user has to type the [filename], see Figure 8, and press the ENTER key on the keyboard.
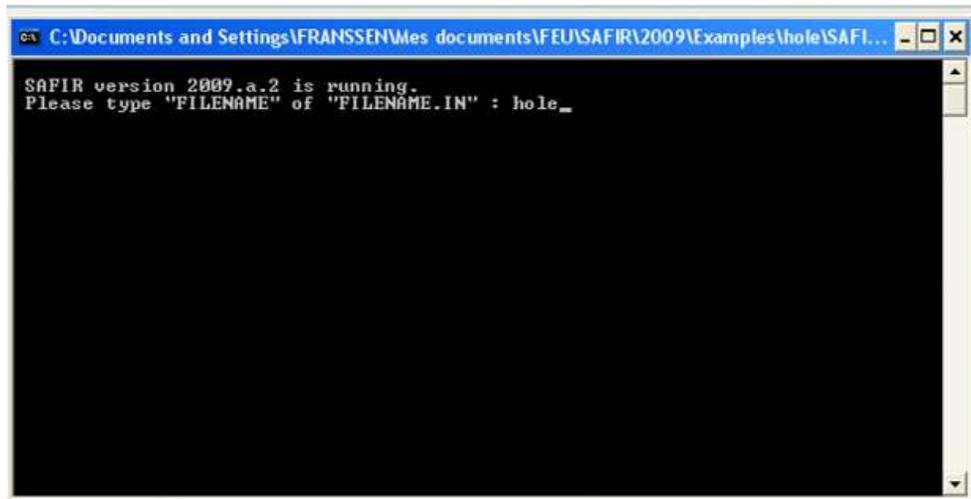
Figure 8: DOS window with the file name

SAFIR starts running and some messages are written in the DOS window, which allows seeing the progress of the simulation, see Figure 9. When the simulation stops, the window is closed immediately[5].

---

[5] If an error message is written as the last line in the window, it is normally not possible to read it before the window closes. The same message is normally written at the end of the output file.

Figure 9: DOS window with the messages

The advantage of this method is that the executable of the version of SAFIR that was used for the project will be present in the archive where the project will be saved after completion and will be available in the future if it proves necessary to come back to the project for any reason.

3.7.2 From the SafirShell tool

SAFIR is copied in one single folder of the hard disk, for example in the folder [c:\program files\SAFIR\2009].

With this method, the file *"identity.key"* must be located **in the root directory** of the device where "safir.exe" has been located, for example in c:

SAFIR will then be run with the tool called "*SafirShell*". This tool is installed by execution of "*tools.msi*" that can be downloaded on the SAFIR web site:
https://www.uee.uliege.be/cms/c_4016387/fr/ueenew-ressources-sur-safir.
When this *SafirShell* is started, a window is opened, see Figure 10.
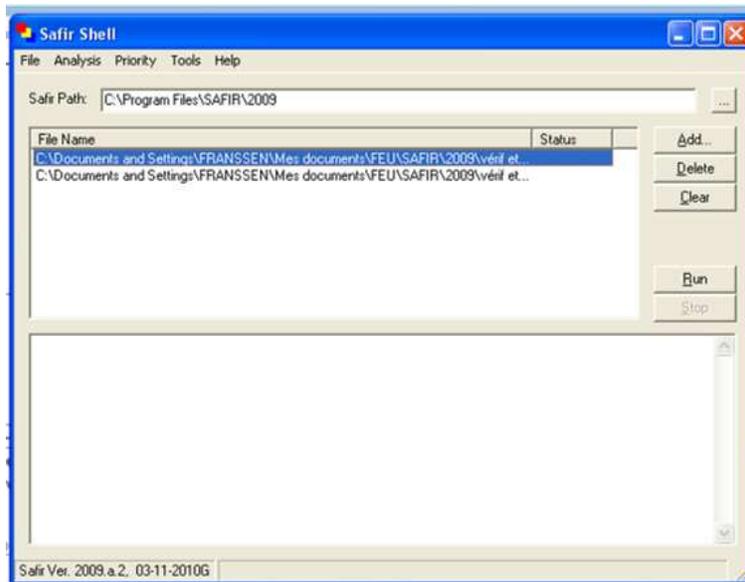
Figure 10: window of SafirShell

The folder where the executable file of SAFIR is located is indicated in the '*Safir Path*' window. This folder can be selected by the user with the […] button on the top right.

The input files that will be treated by SAFIR can be chosen by the user using the [Add..] button. SAFIR will start running as soon as the [Run] button is clicked on.

The advantages of using the SafirShell tool are:

✓ The file "*safir.exe"* is present only in one folder on the hard disk.
✓ Different versions of SAFIR can be located in different folders to be used depending on the choice of the user (to come back to old projects run with older versions, for example).
✓ Several input files can be chosen and treated in a batch mode, one after the other. If one runs fails for any reason, the next input files will be treated.
✓ The messages that are written in the DOS window will now be written in the bottom window of the SafirShell and will remain there to be read even after the runs are finished.
✓ The priority allocated by the Operating System to the SAFIR process can be selected in the 'Priority' scroll down menu. For processors with a single core, it is recommended to choose 'below normal' because this will hardly affect the velocity of SAFIR but will allow time enough to be allocated for performing usual office tasks.

3.7.3 From a BAT file

It is possible to run several successive instances of SAFIR in a BAT file that is launched in a DOS command window.

An example of a BAT file could be:

```
SAFIR    case1_thermal
SAFIR    case1_structural
SAFIR    case2_thermal
SAFIR    case2_structural
```

Since version SAFIR 2022.C.0, it is possible to add two optional arguments at each line of the BAT file.
- ➢ The first argument is always the name of the .IN file. This name is mandatory.
- ➢ The second argument, if present, is the full name of the folder where the .IN, the .OUT and all files processed by SAFIR are located for this run.
- ➢ The third argument, if present, is the full name of the folder where the file "*identity.key*" is located.

```
c:/SafirFolder/safir.exe    case1thermal      c:/InputFolder/  c:/LicenceFolder/
c:/SafirFolder/safir.exe    case1structural   c:/InputFolder/  c:/LicenceFolder/
```

> Note that the second and the third argument finish with a character string "/".

It is possible to have the BAT file, the file "*safir.exe*", the file "*identity.key*" and the files processed by SAFIR located on different devices, possibly linked to different computers. This should facilitate running SAFIR from different terminals connected to a central server.

The files processed by SAFIR and located in the folder described by the second argument are:
- ✓ The IN file
- ✓ The OUT file
- ✓ The XML file
- ✓ The TEM file created in a thermal analysis.
- ✓ The TEM file used in a mechanical analysis. The name of this file given in the input file of the mechanical analysis is the short name, for example "`case1thermal.tem`"; it must not comprise the second argument ("`c:/InputFolder/case1thermal.tem`" would be wrong).
- ✓ The TSH file created in a thermal analysis.
- ✓ The TSH file used in a mechanical analysis. The name of this file given in the input file of the mechanical analysis is the short name, for example "`case1thermal.tsh`"; it must not comprise the second argument ("`c:/InputFolder/case1thermal.tsh`" would be wrong).
- ✓ The user defined function that multiplies the loads in the command LOAD, see Card B in SERIES 23 in "*Part 3: Structural calculations*" of the manual.

The file "*comeback*" which is intensively used in I/O operations during COMEBACK procedures remains in the safir folder as the file "*safir.exe*".

3.8   Typical personal time and setup time to perform a typical run

The typical personal time and setup time to perform a typical run (not counting the time required to gather all information about the structure to be analysed) is in the order of half a day for a 2D analysis and one day for a 3D analysis, although high variations are possible compared to these typical values/

3.9   Estimation of the computer execution time for typical applications

It is not possible to give precise information here as the computer execution time very much depends on the computer on which the execution is made and even more on the complexity of the model. Only orders of magnitude below are given for execution on a decentmy recent laptop.

Thermal calculations.

   2D section for a shell finite element: 1 second

   2D section with some hundreds of nodes: 10 seconds

   2D section with gypsum material: 20 seconds

   2D section with internal cavities: 60 seconds.

   2D sections with localised fires: multiply the number given above by the 2 or 3 times the quantity of BEAM finite elements and/or 4 times the quantity of SHELL finite elements.

Mechanical calculations

   One column or one beam discretised with BEAM finite elements: 10 seconds

   One simple frame modelled with BEAM finite elements: 20 seconds

   One column or one beam modelled with SHELL finite elements: 60 seconds

   One composite steel-concrete slab working in tensile membrane action: 120 seconds

One storey of a car park with 20 slab panels in the floor, plus the beams above and below the floor: 1 800 seconds

4. Program considerations

4.1 Major options available

The first option that the user must take depends on the type of structural calculation that will be performed subsequently after the thermal calculation.

A. A 3D thermal calculation is typically performed to establish the temperature distribution in a detail of a construction, for example a connection, see Figure 11. Such a 3D thermal calculation can be followed by a structural calculation made by SAFIR using 3D solid finite elements. In this case, the discretization made for the thermal calculation is used as is in the subsequent structural calculation. Alternatively, the temperature distribution can be used in a simple structural calculation model such as, for example, the component method in steel connections or the strut and tie model in concrete elements, in which the influence of the calculated temperatures on the materials strength can be considered.
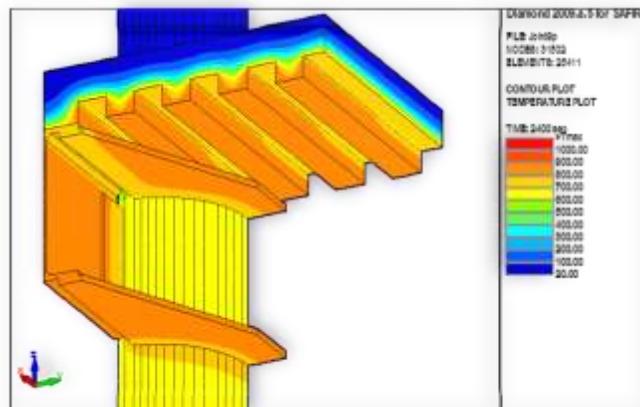

Figure 11: composite steel-concrete joint[6]

B. A 2D thermal calculation can be performed on the cross section of a BEAM finite element, see Figure 12. This means that the discretization made for the thermal calculation will also be used to calculate the mechanical properties of the section in the subsequent structural calculation.

---

[6] Model by Elisabetta Alderighi, Univ. of Pisa.
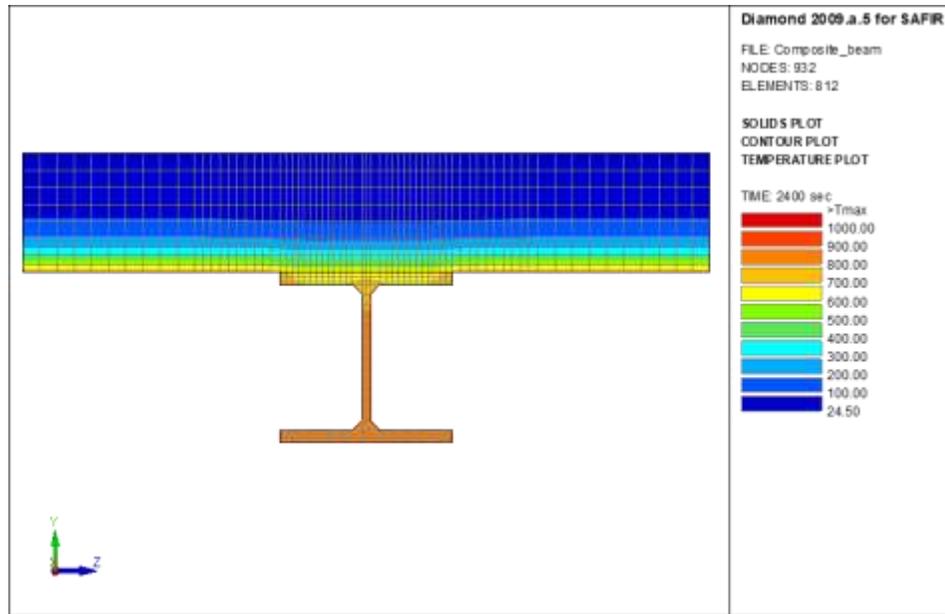
Figure 12: composite steel-concrete beam, isotherm after 2400 seconds

C. A 2D thermal calculation can be performed for evaluating the temperature distribution across the thickness of a shell finite element. Figure 13, for example, shows a 14 cm thick concrete slab protected on the lower side by a 2 cm thick layer of protection material. The discretization must comply with the following rules:

- An arbitrary width of the shell is modelled. In Figure 13, for example, a width of 10 cm has been modelled, but it could be as well 1 cm or 1 meter. Also, the horizontal position of the section in the local system of coordinates is not relevant.
- The section is modelled by rectangular finite elements, the width of which is the same as the width of the discretized section. The quantity of elements on the thickness of the section, N, is chosen by the user (N = 18 on Figure 13)
- The nodes are numbered from 1 in the bottom left corner to N+1 on the top left corner, and from N+2 to 2 N + 2 on the right side of the section.
- In the local system of coordinates, the structural material of the slab is located from -t/2 to +t/2, in the vertical direction, with t the thickness of the structural part of the slab (from node 5 to node 19 on Figure 13).
- There is only one structural material represented in the thermal analysis of the slab (material SILCONC_EN in Figure 13). If re-bars are present in the slab, they are not considered in the thermal analysis but will be added in the ".IN" file of the structural model. Layers of other materials, typically insulating products, can be present outside the structural part of the shell (material INSULATION in Figure 13).

25
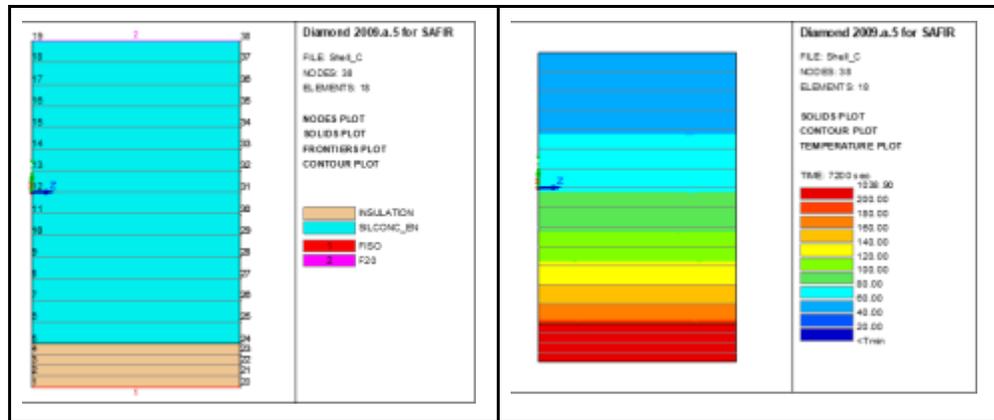
Figure 13: concrete shell protected on the bottom face

Figure 14 shows a 12 mm steel plate exposed to the fire on both sides and modelled by a single finite element. A width of 100 mm has been considered here, but this value does not influence the temperature distribution across the thickness and is thus not relevant.
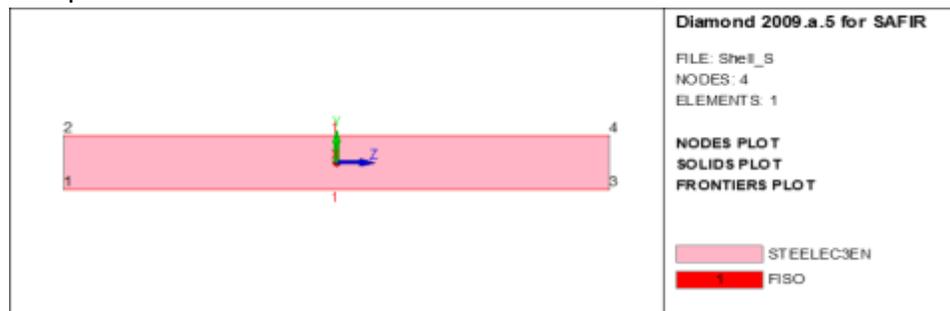
Figure 14: steel plate

The second option to be decided by the user is whether the fire is considered as a localized fire or not.

A. Very often, the fire is not localized. Large parts of the building are considered to be characterized by a uniform situation. For example, one room of the building is characterized by a post flash-over fire curve, while the other rooms are not subjected to the fire. Or a fire compartment is divided in an upper zone that is characterized by an elevated but uniform temperature and a lower zone in which the temperature is lower, but also uniform. In these cases, several finite elements are subjected to the same attack from the fire. It is therefore sufficient to determine the temperature in one of all the elements of the same type which are subjected to the same attack. For example, if a steel beam located in a post flash-over compartment is represented by shell finite elements, it is sufficient to determine the temperature distribution once for the flanges and once for the web[7]. To perform these two thermal calculations, it is sufficient to know the level of the fire attack (the fire curve, for example). It is not necessary to know the position in the compartment of the element for which the temperature is being determined (because all elements of the same type in the compartment have the same temperature distribution).

---

[7] If the flanges and the web have different thickness

B. In some cases, the fire is a localized fire, which means that its dimensions are small compared to the dimensions of the compartment. This could be the case, for example, of one car burning in a large car park. In this case, all finite elements in the compartment are subjected to a different attack from the fire, essentially because the radiative heat transfer from the localized fire to the elements depends on the distance between both and on the angle between the surface of the element and the direction of the fire. It will then be necessary to perform a thermal calculation in a lot of different positions of the structure in order to represent the fact that the temperature distribution varies from places to places. Typically, a uniaxial temperature distribution will be determined across the thickness in 4 different positions of each shell finite element and a 2D temperature distribution will be determined in each of the longitudinal point of integration of each beam-column finite element. In order to perform this series of independent thermal calculation, SAFIR must know where each section is located in the compartment with respect to the fire. This means that the structural model must have been created and the position of characteristics of the fire must have been positioned in the compartment before the thermal calculations can be performed.

Local fire models implemented in SAFIR are the co-called models of HASEMI, LOCAFI, TRAFIR and FDS, see "Manual of SAFIR 2022 - 2) Thermal.docx".

4.2 Limits of applicability

This section presents the range of scenarios over which the underlying theory is known or believed to be valid and the range of input data over which the calculation method was tested.

- The scenario that has been considered here are fire scenarios in building structures, with a duration that extends typically between some minutes and a few hours. Extremely fast heating rates such as those that may occur, for example, in the launch pad of Ariane rockets, are not within the scope of the calculation method used in SAFIR. On the other end of the range, applications that involve very long exposure to elevated temperatures such as, for example, those that may be encountered in some industrial processes, are also not within the scope of SAFIR.
- The fire scenario may include, after the heating phase, a so called cooling phase leading to decrease of the temperature in the materials. Such a cooling phase can be considered by SAFIR as far as material properties during cooling are known.
- The range of temperature is from 0°C to 1200°C. The effect of freezing of free water that may be present in the materials is indeed not taken into account. On the other end of the temperature range, the material properties for fire applications are not known above 1200°C.

4.3 Restrictions and limitations of SAFIR

In this section are listed the restrictions and limitations of SAFIR, including appropriate data range and SAFIR's behaviour when the ranges are exceeded.

Appropriate data range	Behaviour when the ranges are exceeded
SAFIR can treat 2D or 3D models	If *ndim* is not 2 or 3, a message is written in the output file and SAFIR stops execution.
Quantity of nodes must be at least 3	If *nnode* is less than 2, a message is written in the output file and SAFIR stops execution.
teta must be greater than 0 and not greater than 1	If *teta* is out of scope, a message is written in the output file and SAFIR stops execution.
The initial temperature should be greater than 0 and not greater than 1200	If *tinitial* is out of scope, a warning message is written in the output file, but the run continues
The quantity of cores of the computer should be greater than 0	If *ncores* is less than 1, it is changed to 1 by SAFIR.
The quantity of cores mentioned in the input file should not be larger than the quantity of cores of the computer	If *ncores* is larger than the quantity of cores of the computer, the run continues
The quantity of materials must be greater than 0	If *nmat* is less than 1, a message is written in the output file and SAFIR stops execution.
The quantity of solid elements must be greater than 0.	If *nsolid* is less than 1, a message is written in the output file and SAFIR stops execution.
The quantity of integration points in each direction must be 1, 2 or 3.	If *ng* is not 1, 2 or 3, a message is written in the output file and SAFIR stops execution.
The quantity of frontiers in a cavity must be at least 2.	If *nfrontiervoid* is less than 2, a message is written in the output file and SAFIR stops execution.

5. Input data description

5.1   Input variables

5.1.1 General

The data to be used by SAFIR must be placed in a ".txt" file[8].

The name of the file, [filename], is chosen by the user and must conform to the rules of Windows. For example, some characters cannot be used in the name of the file. [filename] is limited to 70 characters.

The file type of the input file must be ".IN". The complete name of the input file in the folder where it is present will thus be [filename.IN]. On the hard disk of a particular P.C., the full name could be, for example if [example1] has been chosen as the file name by the user, [c:/my files/projects/SAFIR/UsersManual/example1.IN]. This input file will be referred to in this manual as the ".IN" file.

In general, each line of the ".IN" file starts with a command; it is a chain of characters typed in capital letters, with a maximum length of 10 characters. This command indicates to SAFIR the meaning of the numbers that are present on the same line or in the subsequent lines after the command. The commands are also useful for the user who reads the file; they make it quite easy to locate particular information in the input file compared to a situation where only numbers would be present in the file.

```
        NNODE    1   2.   3.
```

The format for writing the input of each card is free. This means that the real values can be written in any form. "235.E6", "235000000." and "235.00E+06", for example, are equally valid and express the same value. Subsequent values must be separated by any number of spaces, or by a comma. The values that are described as belonging to a card in the manual can be split on several lines. Additional information (such as characters used to comment the file) can be written after the last information of a card, provided that it is written on the same line as the last information. The following examples are all valid examples of the same card.

```
        NNODE, 1, 2., 3.
```

```
        NNODE    1
                 2.
                 3.
```

```
        NNODE    1
                 2.   3.    These are the coordinates of node 1
```

The order in which the data are introduced in the input file is mandatory. All data must be introduced in the order that is mentioned hereafter.

Blank card can be introduced in the file between the cards mentioned hereafter.

---

[8] Also called an ASCII file.

5.1.2 Units

The international system of units is adopted in the input files. Hence, for instance, all length quantities are in meters, forces are in Newton, masses are in kilograms, time is in seconds and material strength are in Pascal.

Two exceptions are:

➢ temperature is in degree Celsius.
➢ angles are in degrees.

Most output are in the same system of units. Eventual exceptions are clearly mentioned in the output files.

5.2 <u>Sign conventions</u>

5.2.1 Global system of coordinates

Global axes are employed when defining a structure that is to be analysed using SAFIR. This is done using the Cartesian system of coordinates[9].

➢ For 2D problems, the axes are named G1 and G2.

- In 2D thermal analyses, G1 is typically pointing upward whereas G2 is horizontal to the right[10]. This is how Diamond will represent the model.

- In 2D mechanical analysis, G1 is typically horizontal to the right whereas G2 is pointing upward. This is how Diamond will represent the model. Applied force and displacements are positive in the direction of G1 and G2; applied moments and rotations are positive in a counterclockwise direction.

➢ For 3D problem, the dextrorsum global axes are named G1, G2 and G3. Applied force and moments, displacements and rotations are all positive in the G1, G2 and G3 directions. Although gravity is not relevant in SAFIR, Diamond will represented G3 as vertical and upward at opening.

---

[9] Nodal coordinates can be introduced in a cylindrical system of coordinates, but they are transformed thereafter in a cartesian SoC.

[10] This has to do with the local SoC of a 3D beam finite element, where the cross section is represented by a 2D thermal analysis.

5.2.2 Stresses and effects of action

The stresses are positive in tension.

Axial forces, obtained as a summation of the stresses σ_i, are also positive in tension.

Bending moments M_z in the BEAM elements, obtained from, $M_z = \int_{section} \sigma(y - y_O)\,dA$, with y measured on the local axis of the beam L2, are positive when fibres having a positive local coordinate y are in tension.

Shear forces calculated as $V_z = \frac{(M_{z,n} - M_{z,1})}{dL}$ where $M_{z,n}$ is the bending moment at the last longitudinal point of Gauss in the BEAM element, $M_{z,1}$ is the bending moment at the first longitudinal point of Gauss in the BEAM element, and dL is the distance between these two points of Gauss.

Similar for M_y and V_y.

5.2 Input techniques

There are four known techniques for creating input files.

5.2.1 Text editor

Any text editor that allows creating, opening, modifying, and saving text files can be used to create input files.

This technique is the best choice if one or a few input data have to be modified in an existing input file. New input files can easily be created by the '*SaveAs'* option of the editor if the model to be analysed is very close to an existing model.

When an input file has to be created from scratch, or if extensive modifications have to be made in an existing file, it is recommended to use an editor that has sufficient capabilities such as, e.g., working in column mode or comparing files. The developers have been using ultraedit (http://www.ultraedit.com/) with great satisfaction.

5.2.2 Personal programming

Users with programming skills can write their own software or scripts for creating input files. This requires significant efforts for a program that, typically, will allow creating a very specific type of model to be created, with very little, if any, versatility. This technique can be the best choice if a very large quantity of similar models has to be created, for example in order to perform a parametric analysis. This technique could also be considered for a user that has only one type of structure to analyse with varying dimensions.

Any programming technique can be used, as far as it can create a text file. Spreadsheets such as Excel or scripts written in Python have been used with success.

5.2.3 GID

GID (http://gid.cimne.upc.es/) is a commercial pre-processor that can help creating a finite element model with a graphic user interface, see Figure 15.

GID has been used for SAFIR until versions 2019. GID can be used to create meshes and input files for SAFIR2022, but some editing may be required to fit with the format of this version of SAFIR.
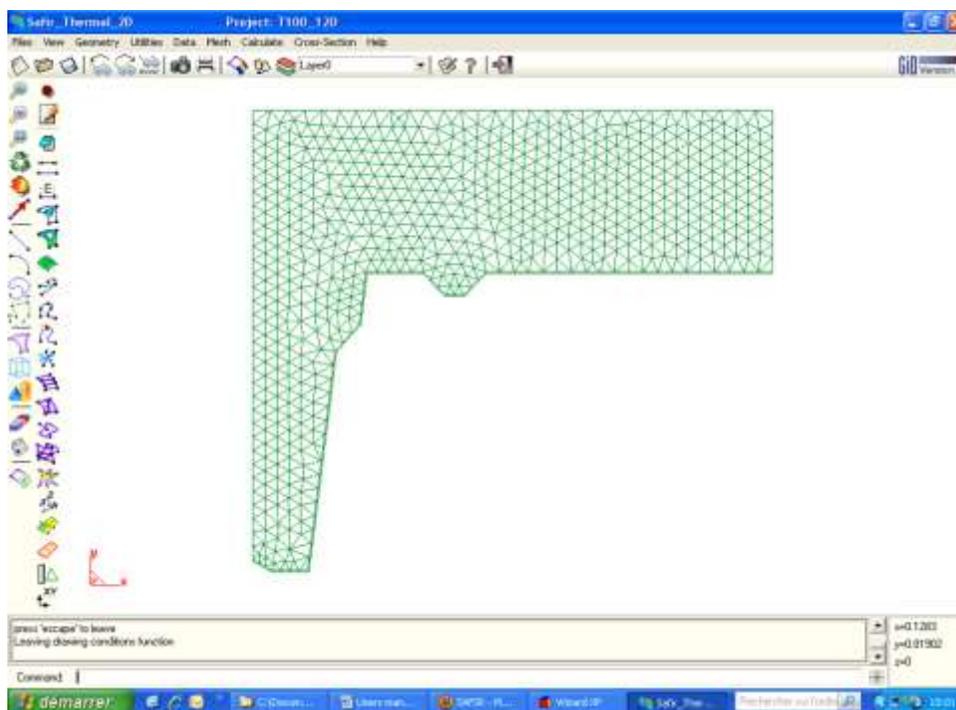
Figure 15: Graphic interface of GID

A reasonably powerful version can be downloaded and run for free, while the full professional version can be purchased with a price that is different for educational organisations or for corporate organisations.

The modules that can make GID writing the input files in the format of SAFIR (for thermal as well as for structural calculations), as well as some short tutorials, can be downloaded on the SAFIR web site (https://www.uee.uliege.be/cms/c_4016387/fr/ueenew-ressources-sur-safir).

5.2.4 GmSAFIR

GmSAFIR is derived from the mesh generator GMSH. Is it free and can be used to generate input files for SAFIR from versions 2022. It is currently under testing and will be available on the SAFIR web site (expected for the end of February 2022).

Mesh generators are clearly the best choice when various and complex models must be created. Experience nevertheless shows that users very quickly learn how to create their models in GID or GmSAFIR, but they have a tendency to let the quantity of nodes and elements increase significantly compared to models made with a text editor. This is not detrimental for single 2D thermal calculations. More care should be exercised and more skills in using mesh generators should be gained to improve the quality of the model when multiple 2D calculations have to be performed because of exposure to a local fire or when really big 3D models have to be analysed.

5.3 <u>Limits on input based on the stability, accuracy and practicality of the data</u>

For thermal calculations, the main limits based on stability and accuracy lay in the link that must exist between the time step used during time integration and the size of the finite elements in the direction of the heat flow.

Accuracy increases when the time step decreases.

Suitable values of the time step depend on the types of material that are present in the model and on the geometry of the model. With usual finite element sizes, in the order of 10 mm perpendicular to the boundary of the model in the zones near the surfaces of the models, times steps from 12 to 20 seconds have been used without any problem in concrete or steel models, protected or not. Higher values can eventually be used after the first 30 minutes of an ISO fire. When gypsum material is all or part of the model, time steps as short as 1 second may have to be used from the very first minutes of the simulation. When the section comprises one or several internal cavities, time steps in the order of 1 second must also be used from the moment when the heat wave reaches the cavities and heat exchanges by radiation in the cavities start playing a role.

In structural calculations, accuracy of the response is linked to the sizes of the elements which are used, with too large elements generally leading to an excessive load bearing capacity of fire resistance time. The size of the time step, the value of the COMEBACK and of the PRECISION may also influence the time when the last converged time step is reached.

5.4 Default variables - Process for setting those variables to user-defined values

There is no default variable in the model other than physical constants.

Default thermal and mechanical material laws are embedded for the materials from the Eurocodes, namely steels, concretes, aluminium's and wood. Default thermal properties are also given for gypsum materials.

The user can introduce its own thermal properties for the materials `INSULATION` and `USERx`.

5.5 SAFIR functions and user defined functions

5.5.1 General principle

In different locations of the [xxx.IN] files, some functions of time can be introduced. They are used either to describe the evolution of the gas temperature or of an applied flux in case of a thermal analysis, or to prescribe the evolution of the solution in different nodes and degrees of freedom (either be it a displacement, a temperature or a value of the warping function).

There are two different types of functions:
- SAFIR defined functions. These functions are embedded in the code. Each function is represented by a name. The comprehensive list of these functions is given in Section 5.5.2.
- User defined functions. If the name of the function (maximum 10 characters) given in the '.IN" file is not one of the SAFIR defined functions, SAFIR will assume that it represents the name of a file in which the user has described the evolution of the function with time by a series of ($time_i$, $value_i$) pairs (free format). Linear interpolation is made between the defined pairs. This file describing the function must be in the same folder as the input file.

5.5.2 SAFIR defined functions

The comprehensive list of SAFIR defined function is (with t in seconds):

```
F0                    f = 0
F1                    f = 1
F1PS                  f = t
MOINSF1PS             f = - t
```

```
FMOINS1PS          f = - t
F2PM               f = 2 t / 60
F20                f = 20
F100               f = 100
F1000              f = 1000
F1000PS            f = 1000 t
F1PSM1000          f = 0                    for t <= 1000
                   f = t – 1000             for t > 1000
F1000THPS          f = t / 1000
FLOAD              f = t / 20               for t <= 20
                   f = 1                    for t > 20
FLOAD2             f = 0.1 t - 0.0025 t²    for t <= 20
                   f = 1                    for t > 20
```

FISO $\quad\quad\quad\quad f = 20 + 345 \log_{10} (8\ t / 60 + 1)$

FISO0 $\quad\quad\quad f = 345 \log_{10} (8\ t / 60 + 1)$

HYDROCARB $\quad\quad f = 20 + 1080\ (\ 1 - 0.325\ e^{-0.167\ t/60} - 0.675\ e^{-2.5\ t/60}\ )$

HCM $\quad\quad\quad\quad f = 20 + 1280\ (\ 1 - 0.325\ e^{-0.167\ t/60} - 0.675\ e^{-2.5\ t/60}\ )$

ASTME119 $\quad\quad$ f defined by linear interpolation between a set of (time ; temperature) pairs, with time in minutes, see Table 1.

RWS $\quad\quad\quad\quad$ f defined by linear interpolation between a set of (time ; temperature) pairs, with time in minutes.

Table 1: set of (time ; temperature) pairs for the ASTME119 function

Time	Temp.	Time	Temp.	Time	Temp.	Time	Temp.	Time	Temp.
0	20								
5	538	65	937	130	1017	250	1100	370	1184
10	704	70	946	140	1024	260	1107	380	1191
15	760	75	955	150	1031	270	1114	390	1198
20	795	80	963	160	1038	280	1121	400	1204
25	821	85	971	170	1045	290	1128	410	1211
30	843	90	978	180	1052	300	1135	420	1218
35	862	95	985	190	1059	310	1142	430	1225
40	878	100	991	200	1066	320	1149	440	1232

45	892	105	996	210	1072	330	1156	450	1239
50	905	110	1001	220	1079	340	1163	460	1246
55	916	115	1006	230	1086	350	1170	470	1253
60	927	120	1010	240	1093	360	1177	480	1260

5.5.3 User defined functions

An example of user defined function could be

```
myfire.fct
```

With a maximum of 10 characters in the name of the file, and the content of the file [myfire.fct] would
be:

```
    0.     20.
  600.    200.
  720.    800.
 2400.    900.
 3600.    300.
 7200.     20.
10800.     20.
```

for a natural fire reaching a maximum temperature of 900°C after 40 minutes and decreasing
thereafter.

5.6   Consecutive cases

Consecutive cases can be executed one by one, if the user starts "*safir.exe*" directly from the
Windows Explorer, or automatically either by the SafirShell or in a BAT file.

In any case, to the knowledge of the developers, all variables are completely re-initialized; there
is no data retention from case to case. Any observation of the contrary should be reported as a
bug to the developers.

6. External data files

There is no external data file used by SAFIR.

SAFIR only uses the files required for the input, see sections 2, 3 and 4 of this manual.

7. Terms and definitions

Card

A series of information of the input file that is read by in single READ statement by SAFIR. The rules for reading a card are the rules of the FORTRAN language. The input data of a single card can be split in the input file on several consecutive lines, see section 4.1.1.

Discretization

The process by which the user represents the structure (that, by nature, is a continuous object) into a model made of a finite quantity of nodes and elements.

Iteration

Because the problems to be solved are nonlinear, SAFIR cannot calculate the solution (either temperatures or displacements) directly at a given time step as a function of the solution that has been calculated at the previous time step. To calculate the solution at a given time step, SAFIR has to perform several trial-and-error calculations during which the solution is progressively refined. Each of these trials and errors is called an iteration.

Line

A physical record of the input file.

Point of Integration

A point in the structure where the material properties are evaluated. The properties at all points in a finite element are used in the numerical integration made on the volume of the element to compute the conductivity matrix and the thermal fluxes (for a thermal analysis) or the stiffness matrix and the nodal forces (for a mechanical analysis) of the element and hence, after assembly of the contribution from all elements, the corresponding matrix, forces or fluxes of the structure.

Time step

A particular time during the fire for which SAFIR will calculate the solution (times steps used for the calculation). The time steps are chosen by the user if NOCOMEBACK has be used.

In case of COMEBACK, the time step is automatically adapted by SAFIR. The user introduces the value of the first timestep, as well as the maximum and the minimum values that the time step can reach.

User

The person who creates a numerical model representing the problem to be solved that can be analysed by SAFIR and who interprets the results provided by SAFIR.