SAFIR®

General Presentation of the software SAFIR®

Capabilities and Examples of Applications

Jean-Marc Franssen (Liege University) & Thomas Gernay (Johns Hopkins University)

SAFIR®

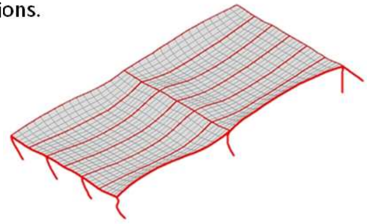
Table of content

SAFIR®

Description of SAFIR®

SAFIR is a **computer program** that models the **behavior of building structures subjected to fire**. The structure can be made of a 3D skeleton of linear elements such as **beams and columns**, in conjunction with planar elements such as **slabs and walls**. Volumetric elements can be used for analysis of details in the structure such as **connections**. Different materials such as **steel, concrete**, **timber**, **aluminum**, **gypsum** or thermally insulating products can be used separately or in combination in the model.

It is used for **research** and for **commercial** applications.



2018 3

SAFIR®

3 steps procedure in a SAFIR analysis

1. The **thermal attack from the fire** is given as an **input data**
2. SAFIR computes the **evolution of temperature** in the sections (thermal analysis)
3. SAFIR computes the **mechanical behavior** of the structure at elevated temperatures, taking into account the thermal elongations as well as the reduction of strength and stiffness in the materials (mechanical analysis)

The numerical analyses are based on the non linear **finite element method**.

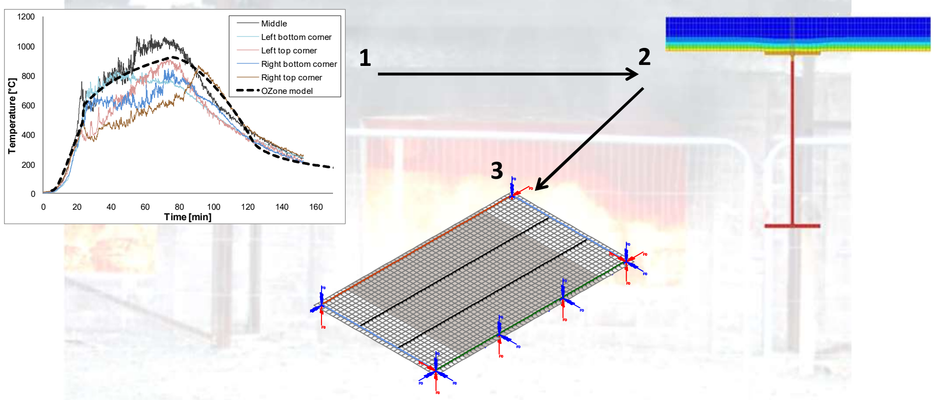
- 2D or 3D conductive elements for thermal calculations
- Linear elements for modelling beams, columns *(Bernoulli beam type)*
- Plane elements for modelling slabs, walls, steel plates (*shell type*)
- 3D volumetric F.E. for modelling connection details, massive members (*3D solid type*)

2018 4

SAFIR®

3 steps procedure in a SAFIR analysis

Example: numerical simulation of the FICEB full-scale fire test

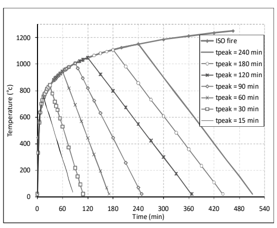


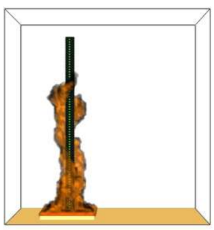
Vassart et al. (2012). "Large-scale fire test of unprotected cellular beam acting in membrane action". *Structures and Buildings, 165*(7), 327–334

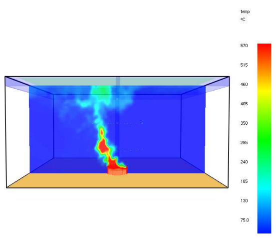
SAFIR®

Step 1: Define the fire

- The thermal action produced by the fire is given as an input data by SAFIR.
- The thermal action can be represented by various methods:
  - Time-temperature curve (standard fires or user-defined curves)
  - Imposed flux
  - Local model from a local fire to a beam or ceiling (Annex C of EN 1991-1-2)
  - Local model from a local fire to a column (RFCS project "LOCAFI")
  - Environment calculated from a CFD software (e.g. FDS)







Step 2: Thermal analysis

- SAFIR performs the transient thermal analysis to determine the temperature distribution in the structure
    - 2D or 3D thermal calculations
    - Finite elements: triangular, quadrangular, prismatic (6 or 8 nodes)
    - Transient calculation (temperature varies with time)
    - Predefined thermal material models from Eurocodes proposed: concrete, steel, wood, aluminum, gypsum
    - Possibility to introduce other materials by specifying their thermal properties (either constant or temperature dependent)

Steps 2 and 3: Thermal and mechanical analyses

- Link between thermal and mechanical analyses

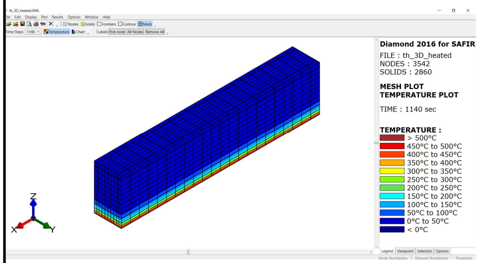
The type of model used for the thermal analysis depends on the type of model that will be used in the subsequent mechanical analysis

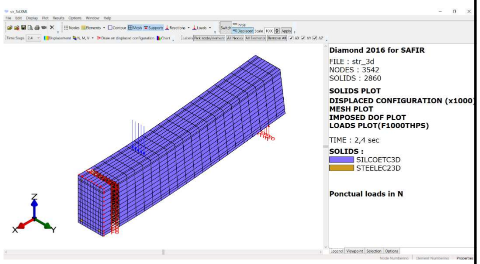
Temperature field	Mechanical model
3D F.E. =>	3D F.E. (only for details)
2D F.E. =>	Beam F.E. (2D or 3D)
1D F.E. (pseudo 2D) =>	Shell F.E. (3D)
Simple calculation model =>	Truss F.E. (2D or 3D)

SAFIR®

Steps 2 and 3:Thermal and mechanical analyses

- For **3D solid elements**, the same discretization is used for the thermal and mechanical analyses so that the **temperatures are directly mapped** on the mechanical model.

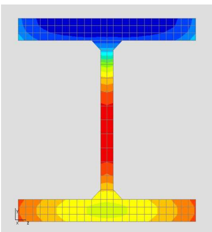


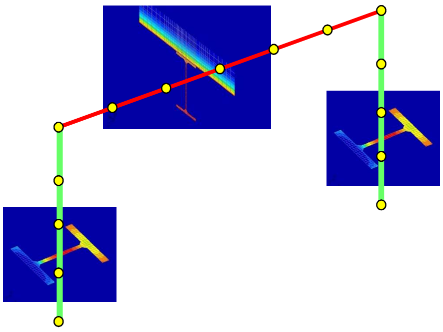


SAFIR®

Steps 2 and 3:Thermal and mechanical analyses

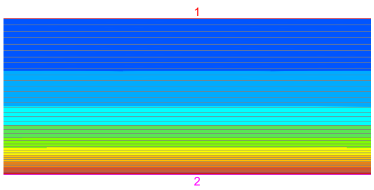
- For **beam elements**, the discretization of the section employed for the thermal analysis (calculation of the temperature at each node) is used in the form of **fibers** for the beam elements in the mechanical analysis. Thus, the determination of forces and stiffness in the section is based on the **temperatures in each element used in the thermal analysis which form a fiber** in the beam element.

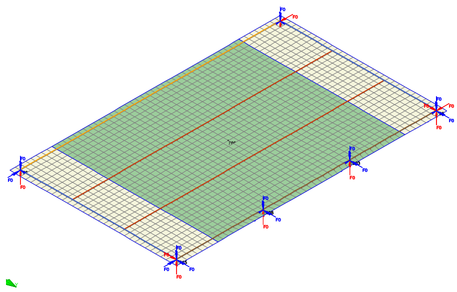




Steps 2 and 3: Thermal and mechanical analyses

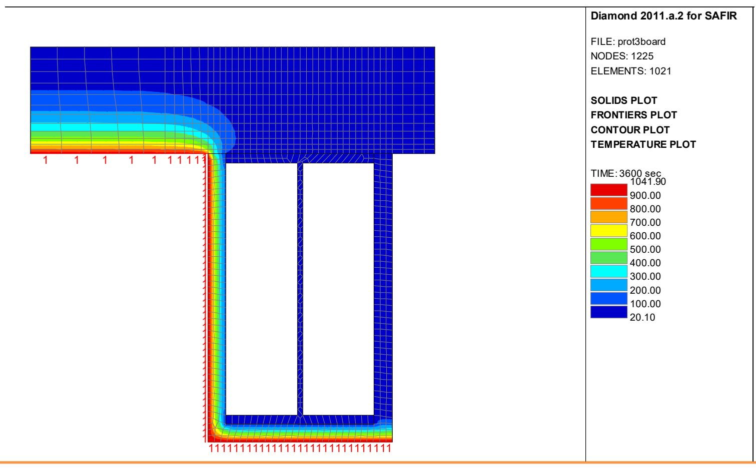
- For **shell elements**, a **uniaxial temperature distribution** is calculated **across the thickness** of the slab using pseudo-2D conductive finite elements. The temperature at the through thickness points of integration for the mechanical analysis is linearly interpolated between the nodal temperatures.





Example of Step 2: Thermal analysis

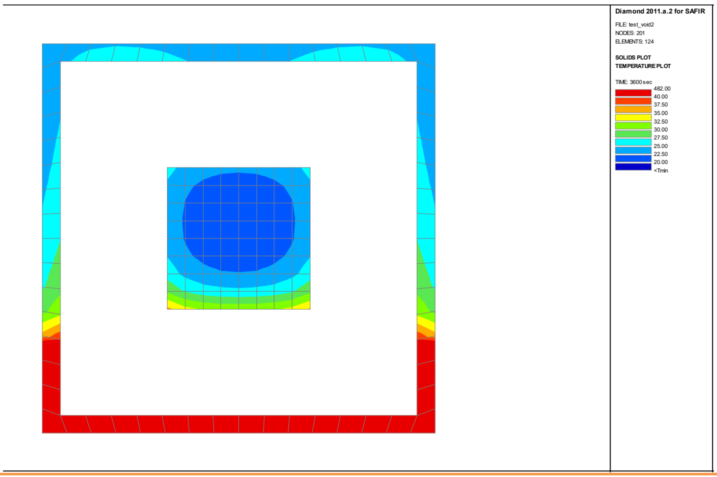
2D thermal calculation - Protected steel beam heated on one side
1 225 nodes - 1 021 quadrangular elements



SAFIR®

Example of Step 2: Thermal analysis

2D thermal calculation – Radiation in internal cavity – shadow effect
201 nodes - 124 elements

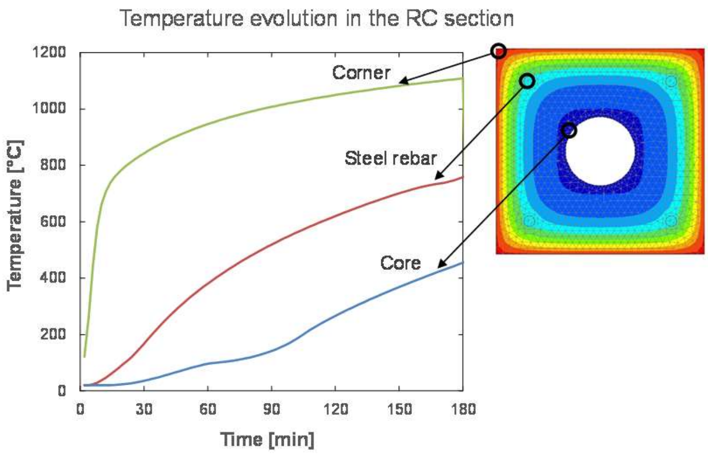


2018 13

SAFIR®

Example of Step 2: Thermal analysis

2D thermal calculation - Reinforced concrete column with hollow core
1097 nodes - 2012 triangular elements



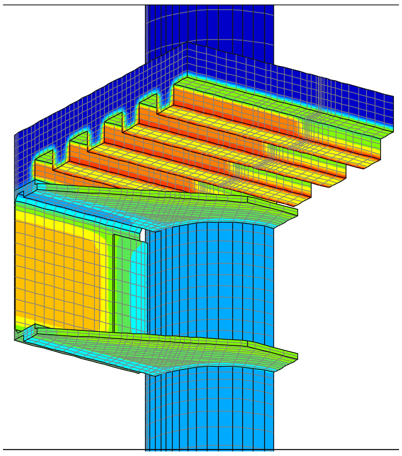
2018 14

SAFIR®

Example of Step 2: Thermal analysis

3D thermal calculation - Composite steel-concrete joint
31 502 nodes - 25 411 solid elements

Full conductivity matrix:	992 376 004 elements
Symmetry used:	496 203 753 elements
Skyline storage:	21 807 326 elements
Sparse matrix solver:	396 963 elements



2018 15

SAFIR®

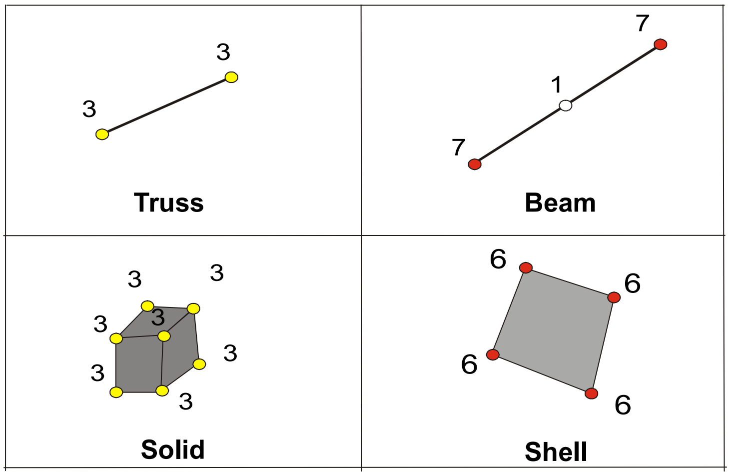
Step 3: Mechanical analysis

- SAFIR performs the transient mechanical analysis to determine the response of the structure (displacements) under increasing temperatures
- It takes into account the effects of thermal expansion and material degradation
  - 2D or 3D structural calculations
  - Finite elements: truss, **beams**, **shell**, solid, spring
  - Nonlinear mechanical properties that are temperature dependent
  - Large displacements
  - Predefined non linear mechanical models of Eurocodes proposed: concrete, steel, wood, aluminum
  - Gives result as a function of time: displacements of the nodes, support reactions, stresses, tangent modulus, effects of actions (M, N, V), etc.
- SAFIR also calculates the torsional stiffness of the section (LTB check)

2018 16

SAFIR®

Step 3: Mechanical analysis - Discretization



SAFIR®

Step 3: Mechanical analysis - Truss FE

- One single point of integration (i.e. one material, temperature and stress level)
- 3 DoF at the two end nodes (translations)
- Cannot represent buckling
- Use:
  - External prestressing tendons
  - Individual rebars in 3D solid elements
  - Bar in tension in a structure (e.g. bracing bar in a building)
  - To create a linear relationship between two nodes

SAFIR®

Step 3: Mechanical analysis - Beam FE

- Prismatic straight Bernoulli type element
- 7 DoF at the two end nodes: translations, rotations, warping
- 1 DoF at the central node to bear the nonlinear part of the axial displacement
- Integration on the section is based on a fiber model
- Longitudinal integration is performed numerically using 2 or 3 points of Gauss
- Warping function and torsion stiffness calculated based on thermal analysis discretization
- Use:
  - Linear members: beams, columns
  - Bars in truss girders (to capture buckling)
  - Steel studs in composite steel-concrete members
  - Semi-rigid connections (taking advantage of fiber model)

SAFIR®

Step 3: Mechanical analysis - Shell FE

- Quadrangle based on 4 nodes
- 6 DoF at each node: 3 translations and 3 rotations
- Integration on the section is performed numerically using 4 points of Gauss
- Integration on the thickness is performed numerically with the user choosing the number of Gauss points (from 2 if membrane behavior dominates to 10 if bending dominates)
- Possibility to embed layers of reinforcement (smeared laterally, uniaxial behavior)
- Use:
  - Planar members: slabs, walls
  - Plates of steel members (to capture local buckling)

SAFIR®

Step 3: Mechanical analysis - Solid FE

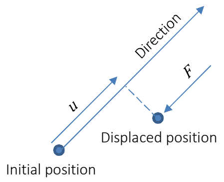
- Based on 6 or 8 nodes
- 3 DoF at each node (translations)
- The user can select from 1 to 3 Gauss integration points in each direction
- Only the quasi-static procedure is available, large displacements not taken into account
- Use:
  - Joints
  - Hollow core slabs
  - Concrete masses

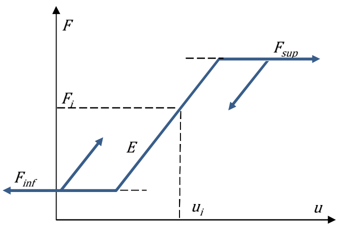
2018 21

SAFIR®

Step 3: Mechanical analysis - Spring FE

- One single node (pertaining to the structure) and one direction
- Its behavior is directly described by a force-displacement relationship (no material)
- Use:
  - To link the structure to the external world via a non-linear relationship
  - Soil pressure on the walls and under the foundations of tunnels
  - Soil pressure on vertical walls of underground car parks



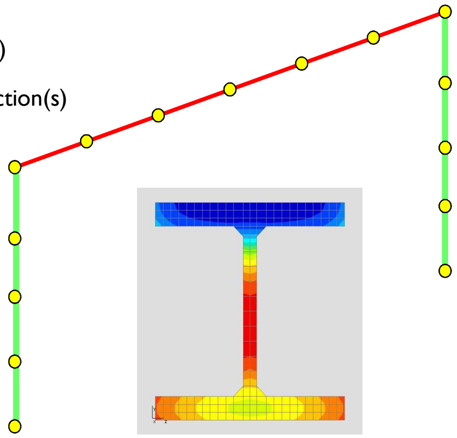


2018 22

SAFIR®

General principle of a mechanical analysis based on beam FE (1/2)

1. Place some nodes in the global system of coordinates
2. Link them with beam elements
3. Define the geometry of the section(s)
4. Calculate the temperatures in the section(s)

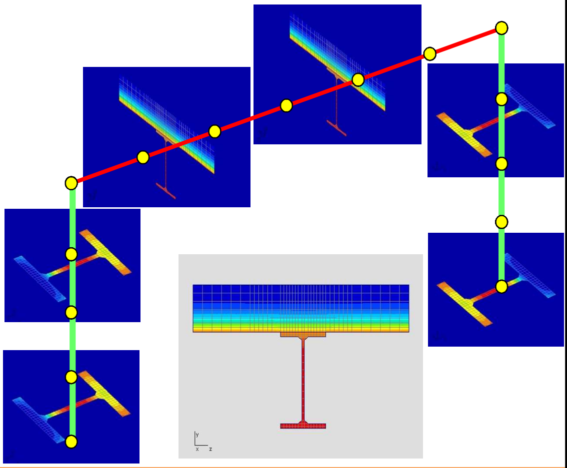


2018 23

SAFIR®

General principle of a mechanical analysis based on beam FE (2/2)

5. Link the section(s) with the elements
6. Define supports and loads
7. Let the heating go

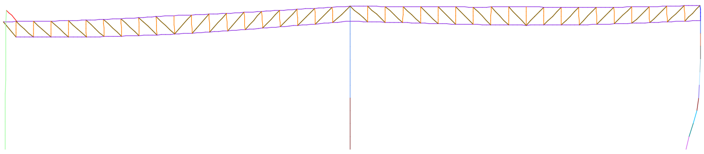


2018 24

SAFIR®

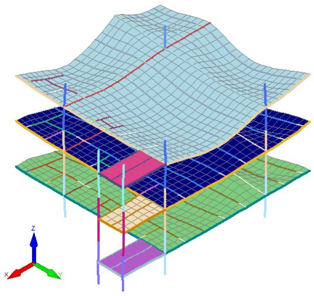
Example of Step 3: Mechanical analysis

2D mechanical calculation – Steel frame



3D mechanical calculation – Steel-concrete building

Case study by R. Fike and V. Kodur
Michigan State University, USA
Partial model of an eight story steel frame office building



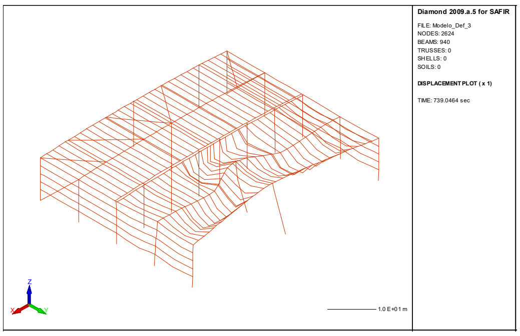
2018 25

SAFIR®

Example of Step 3: Mechanical analysis

3D mechanical calculation – Steel frame

Flumilog test, INERIS, France
2 624 nodes, 940 beam elements



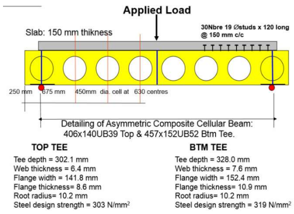
2018 26

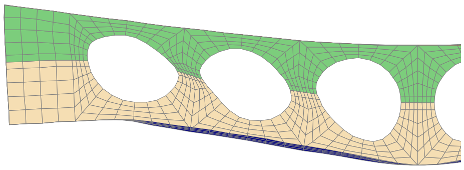
SAFIR®

Example of Step 3: Mechanical analysis

3D mechanical calculation – Non symmetrical composite cellular steel beam in fire

Experimental test and finite element model



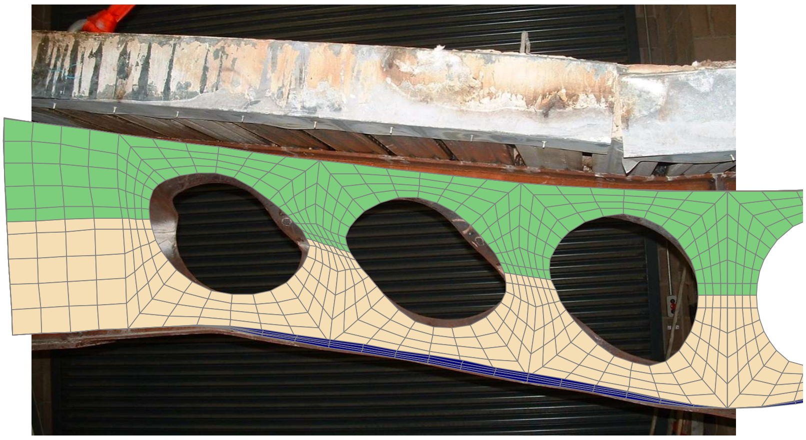


SAFIR®

Example of Step 3: Mechanical analysis

3D mechanical calculation – Non symmetrical composite cellular steel beam in fire

Experimental test and finite element model

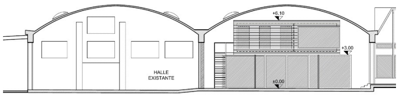


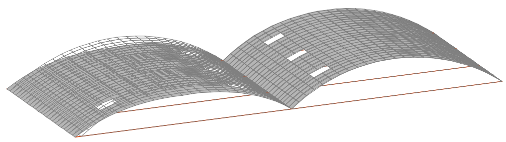
SAFIR®

Example of Step 3: Mechanical analysis

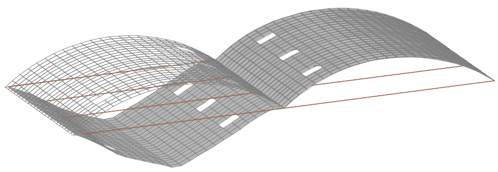
3D mechanical calculation – Structural fire analysis of an arched reinforced concrete roof

Study for ICB, Belgium





Snap-through collapse under fire
when the steel tie rods yield

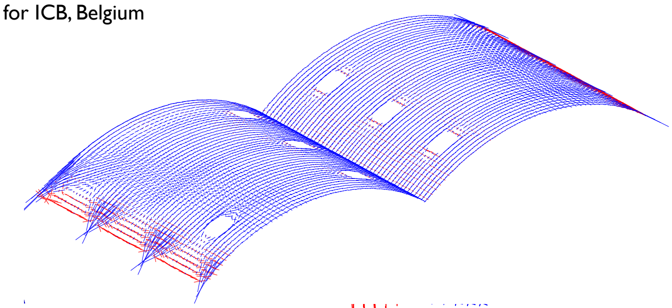


SAFIR®

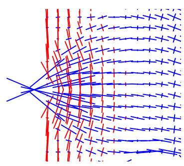
Example of Step 3: Mechanical analysis

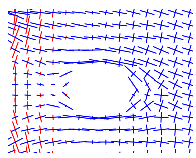
3D mechanical calculation – Structural fire analysis of an arched reinforced concrete roof

Study for ICB, Belgium



Plot of the membrane forces in
the concrete shells

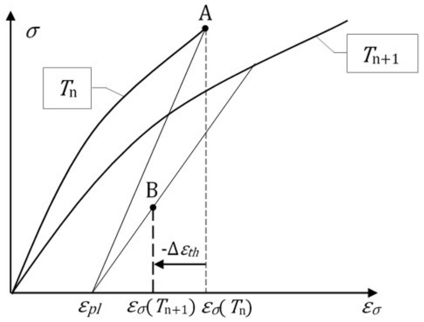




SAFIR®

Step 3: Mechanical analysis – Time integration procedure

- Iterative procedure to integrate in time from one converged time step n (point A) to the next step n+1.
- At converged time step n, the stress-related strain is $\varepsilon_\sigma(T_n)$ and the plastic strain ε_{pl}.



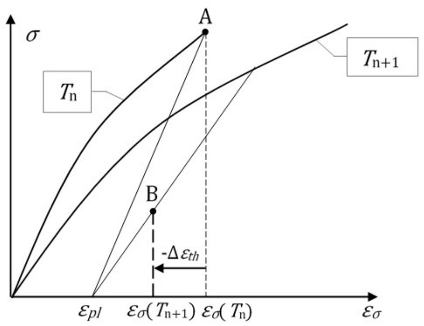
1. The increment of ε_{th} at every PI is calculated based of temperature increments from *n* to *n*+1.
2. The properties of the materials are updated corresponding to T_{n+1}. As a result, the virgin stress-strain law (σ - ε_σ) is different at temperature T_{n+1} compared with T_n.
3. Plastic strains ε_{pl} at the PI as well as nodal displacements are kept constant at the beginning of the step.

2018 31

SAFIR®

Step 3: Mechanical analysis – Time integration procedure

- Iterative procedure to integrate in time from one converged time step n (point A) to the next step n+1.
- At converged time step n, the stress-related strain is $\varepsilon_\sigma(T_n)$ and the plastic strain ε_{pl}.



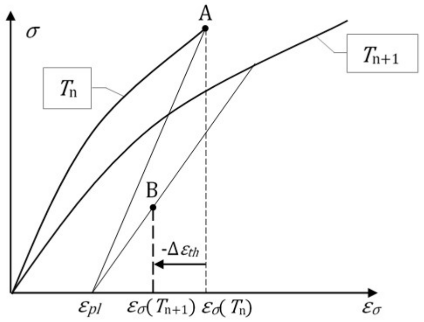
4. With displacements "frozen", ε_{tot} are constant. So the $\Delta\varepsilon_{th}$ generates a new value of stress-related strain, $\varepsilon_\sigma(T_{n+1})$. For a temperature increase, the structure is "artificially compressed".
5. Taking into account $\varepsilon_\sigma(T_{n+1})$, the stress-strain 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).

2018 32

SAFIR®

Step 3: Mechanical analysis – Time integration procedure

- Iterative procedure to integrate in time from one converged time step n (point A) to the next step n+1.
- At converged time step n, the stress-related strain is $\varepsilon_\sigma(T_n)$ and the plastic strain ε_{pl}.



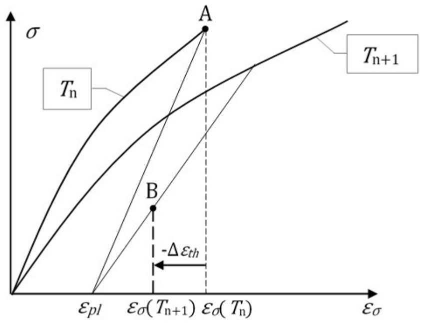
6. The stresses are integrated on the volume of the elements to compute the internal nodal forces which are not anymore in equilibrium with the applied nodal forces. In case of temperature increase, the structure undergoes an internal state of out of balance compression.
7. The stiffness at the integration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

2018 33

SAFIR®

Step 3: Mechanical analysis – Time integration procedure

- Iterative procedure to integrate in time from one converged time step n (point A) to the next step n+1.
- At converged time step n, the stress-related strain is $\varepsilon_\sigma(T_n)$ and the plastic strain ε_{pl}.



8. The out of balance forces are applied to the structure, leading to incremental displacements, new strains (stress related component), new stresses and new nodal forces.
9. The procedure is repeated several times at constant temperature until the convergence criteria is satisfied.
10. Plastic strains are updated after convergence.

2018 34

SAFIR®

Materials available

	THERMAL ANALYSIS		STRUCTURAL ANALYSIS		
Type of FE	2D Solid	3D Solid	Beam Truss	Shell	3D Solid
Type of law			Uniaxial	Plane stress	Triaxial
Mapped with	Beam Shell	3D Solid			
Material:					
Steel	X	X	X	X	X
Concrete	X	X	X	X	X
Wood	X	X	X		
HSC	X	X	X		
Stainless steel	X	X	X		
Aluminum	X	X	X		
Gypsum	X	X			
Insulation	X	X			
User	X	X			
User_Steel			X		

2018 35

SAFIR®

Pre-processor GiD

- GiD allows the generation of any input file for 2D or 3D, thermal or structural problem
- GiD is a universal commercial software developed independently to SAFIR, http://www.gidhome.com/
- The *problem types* are developed by the SAFIR team to allow generating SAFIR input files from GiD: http://www.uee.uliege.be/cms/c_2673990/fr/safir-free-downloads

Post-processor DIAMOND

- DIAMOND allows visualizing the structure and the results
- Allows plotting charts for many results and exporting them to Excel
- DIAMOND is developed by the SAFIR team and can be obtained at: http://www.uee.uliege.be/cms/c_2673990/fr/safir-free-downloads

2018 36

SAFIR®

Organization of the files for a typical calculation – Example for:

- One mechanical calculation for a structure made of beam elements
- In which there are 2 different section types

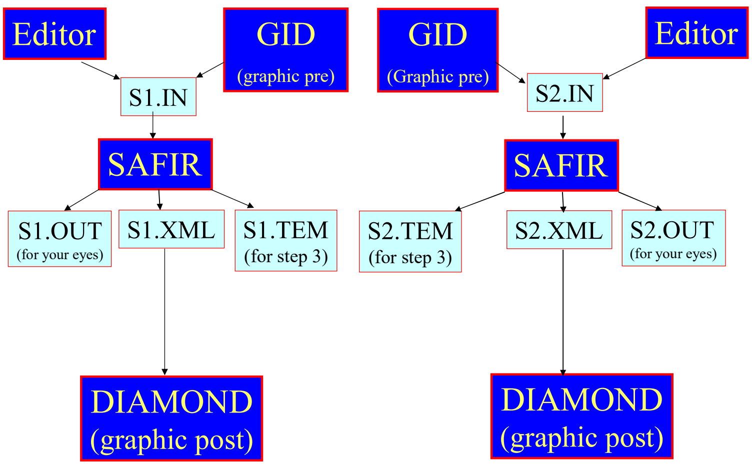
Note: one new section type must be considered if:

- the geometry of the section is different,
- the fire curve is different,
- the thermal properties are different,
- the mechanical properties are different.

  Note: in this latter case, it is possible to copy the results of the thermal calculation in a file with a new name.

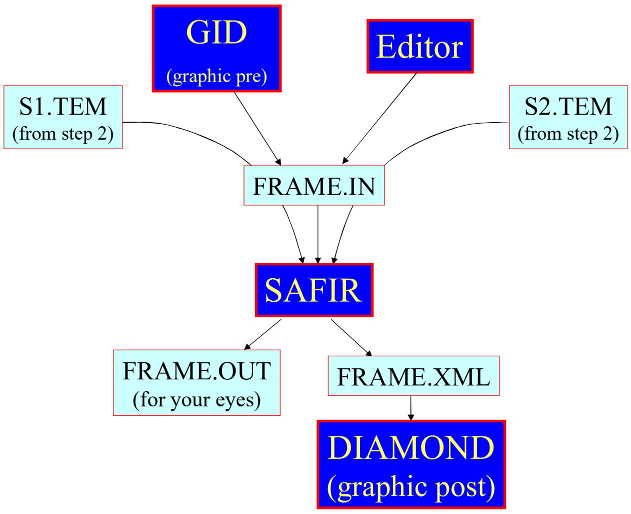
SAFIR®

Organization of the files for a typical calculation – Example, Thermal Analysis



SAFIR®

Organization of the files for a typical calculation – Example, Mech. Analysis



2018 39

SAFIR®

How is failure considered in SAFIR (in structural fire engineering software…) ?

- It is not, because the notion of failure is arbitrary
- Failure can only be defined by (human) interpretation of the results, with different criteria for different structures, different situations and objectives, …

So in practice, how to assess failure from an advanced (numerical) analysis?

2018 40

SAFIR®

So in practice, how to assess failure from an advanced (numerical) analysis?

Case I: the software does not run

⇒ The model needs to be checked for:

- Material properties (f_y or E too small or = 0)
- Boundary conditions
- Mechanism in structures (e.g. axial rotation in diagonals)
- Isolated nodes (nodes not linked to any element)
- Load level (load is too big)
- Time step (too big)
- …

Case II: the software runs

⇒ Examine the results

SAFIR®

So in practice, how to assess failure from an advanced (numerical) analysis?

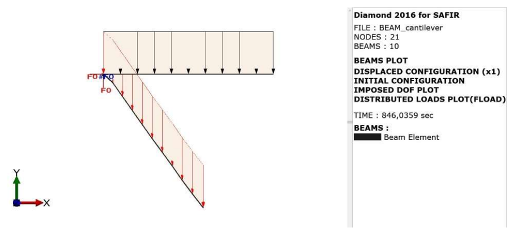
Case II: the software runs

⇒ Examine the results

A. The displacements are **too big** (from my point of view)

⇒ I decide that fire resistance is lost when the displacements reach my limit

- Horizontal displacement at support leads to loss of support of the beam
- Beam of a frame deflects into the ground
- Cantilever beam is transformed into a cable hanging on the support



SAFIR®

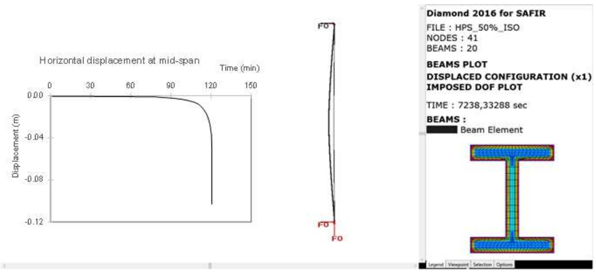
So in practice, how to assess failure from an advanced (numerical) analysis?

Case II: the software runs

⇒ Examine the results

B. The displacements are not too big

⇒ Look for vertical asymptote in the displacement curve of one DoF, which is a good indication of runaway failure



SAFIR®

So in practice, how to assess failure from an advanced (numerical) analysis?

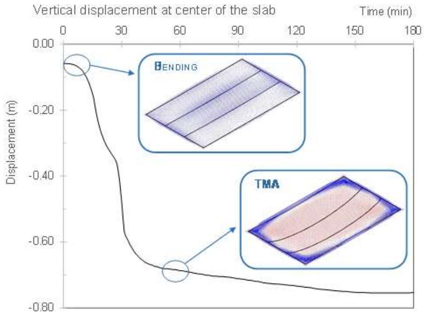
Case II: the software runs

⇒ Examine the results

B. The displacements are not too big

⇒ Look for vertical asymptote in the displacement curve of one DoF, which is a good indication of runaway failure

Note: exception when the failure mode is changing but post-critical behavior is possible. Example: concrete slab going from bending to tensile membrane action; snap through; …



SAFIR®

So in practice, how to assess failure from an advanced (numerical) analysis?

Case II: the software runs

⇒ Examine the results

C. The displacements are not too big, no vertical asymptote

⇒ This is a good indication of numerical failure (premature lack of convergence)

Note: except when the failure mode is really fragile

- Look for material failure (strains, stresses)
- Do a dynamic calculation, more robust
- Change the calculation parameters: time step, numerical strategy, etc.
- Run some alternative analyses to get a better insight into what is causing the problem. For instance, try with an elastic material, without thermal expansion, lower loads, etc.

2018 45

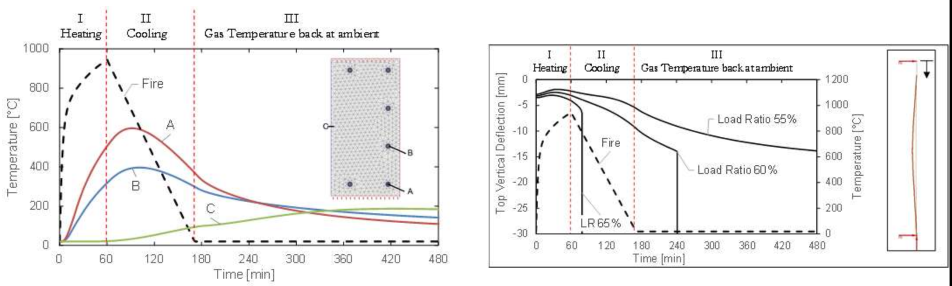
SAFIR®

So in practice, how to assess failure from an advanced (numerical) analysis?

Case II: the software runs

⇒ Examine the results

Note: under natural fire, make sure to run the simulation long enough to detect possible failure in cooling (or even after)!



Gernay, T., & Dimia, M.S. (2013). *Engineering Computations*, *30*(6), 854-872.

2018 46

SAFIR®

Users

SAFIR®

in the world

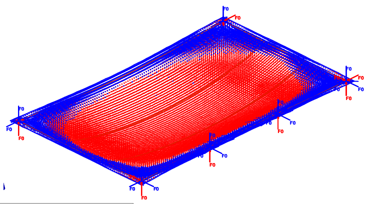
Non linear finite element software for structures in fire

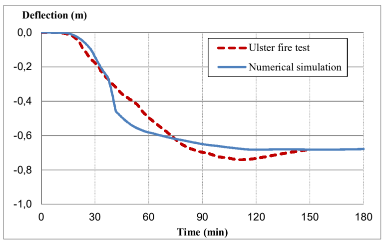
+200 licenses sold
43 countries
5 continents

SAFIR®

Example of scientific reference

Vassart et al., RFCS project FICEB – *Modeling of a full-scale tensile membrane fire test*



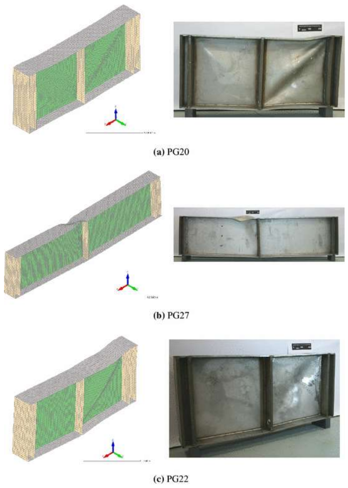


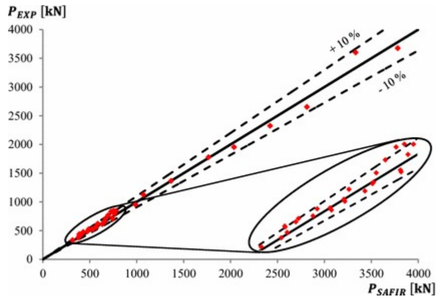
Vassart et al. (2012). *Structures and Buildings, 165*(7), 327–334

SAFIR®

Example of scientific reference

Reis et al. – *Shear buckling of plate girders at normal and high temperature*



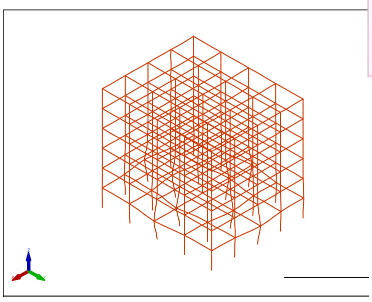


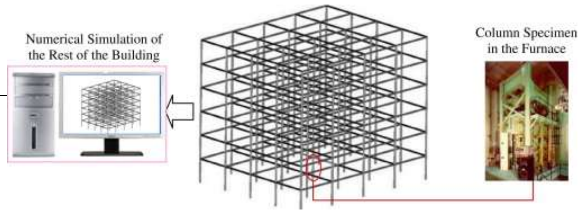
Reis A., et al. (2017) *Fire technology* 53.2: 815-843.

SAFIR®

Example of scientific reference

Mostafaei et al., NRC - *Hybrid Fire Testing for Performance Evaluation of Structures in Fire*





Use of SAFIR for HFT:

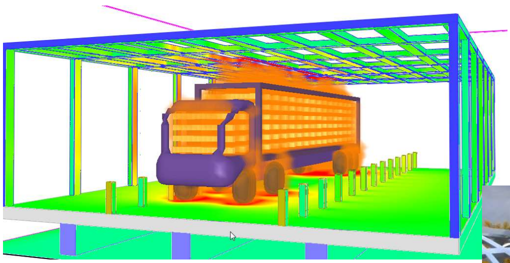
Mostafaei, H. (2013). *Fire safety journal*, *56*, 30-38.

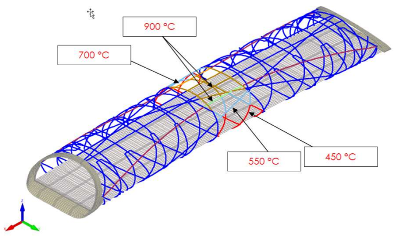
Sauca A., et al. (2018). *Journal of Structural Fire Engineering*.

SAFIR®

Example of commercial reference

MP Ingénieurs Conseils, Switzerland – Wilsdorf bridge in Geneva



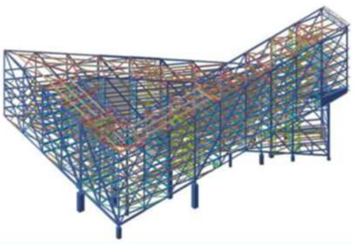


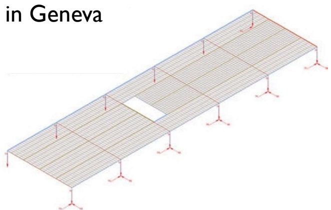
Tonicello, E. et al. (2012). *Proc. 7th Structures in Fire Conf.*

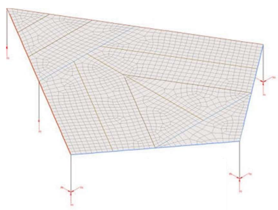
SAFIR®

Example of commercial reference

INGENI, Switzerland – Japan Tobacco Headquarter in Geneva







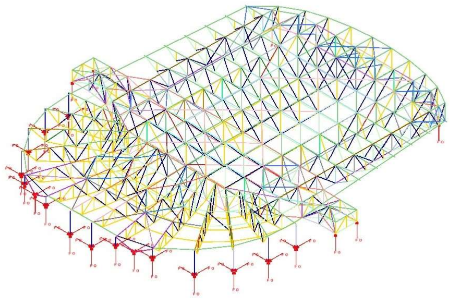
Lelli, L., Loutan, J. (2017). *Journal of Structural Fire Engineering.*

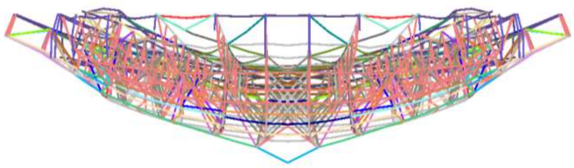
SAFIR®

Example of commercial reference

BuroHappold Engineering, UK

Del Prete, I., Block, F. (2017). Proc. of *IFireSS 2017*, Naples, Italy.





SAFIR®

Validation

- Ferreira, J., Gernay, T., & Franssen, J.M. (2018). Discussion on a systematic approach to validation of software for structures in fire. Structures in Fire (Proc. of the 10th Int. Conf.). Ulster University, UK, Jun 6-8.
- 441 references found in the literature with simulations made with SAFIR
  - 96 provide comparisons between SAFIR and experimental tests
  - 22 provide comparisons between SAFIR and other software
  - Results have been reviewed and classified per typology, fire exposure, etc.
- Modeling of the validation examples in Annex CC of DIN EN1991-1-2/NA(2010)
  - Open data: http://hdl.handle.net/2268/208197
- The full report is at http://www.uee.uliege.be/cms/c_2673990/fr/safir-free-downloads

SAFIR®

Scientific references

- Franssen, J. M., & Gernay, T. (2017). Modeling structures in fire with SAFIR®: Theoretical background and capabilities. *Journal of Structural Fire Engineering*, *8*(3), 300-323. http://hdl.handle.net/2268/202859
- The former journal reference (AISC publication from 2005) has 430 citations on Google Scholar as of May 2018, plus the manual has around 160 citations

Resources

- Manuals, worked examples and input files can be downloaded for free
- Manuals: http://www.uee.uliege.be/cms/c_2673990/fr/safir-free-downloads
- Examples: http://www.uee.uliege.be/cms/c_2674002/fr/safir-applications-examples
- Contact us at safir@uliege.be

2018 55

SAFIR®

Conditions

Licenses: visit https://www.gesval.be/catalogue/safir-2016-commercial

✓Academic license: 1000 € + taxes.

✓Commercial license: 5000 € + taxes.

✓No limitation in time (the license can be used for unlimited duration)

✓One license is valid for multiple users (from a same institution and a same location/site)

✓Free updates during 1 year

Training sessions

✓Organized regularly at Liege University and JHU

✓Organized on demand

- 800 € per day, independent on the number of participants
- Can be organized on site at the client's. In this case, the client also covers the travel and accommodation cost, plus one day at the rate of 800 € for travel time.

2018 56

Thank you!

General Presentation of the software SAFIR®

Franssen, J. M., & Gernay, T. (2017). Modeling structures in fire with SAFIR®: Theoretical background and capabilities. *Journal of Structural Fire Engineering*, *8*(3), 300-323.