SAFIR®

A software for modeling the behavior of structures subjected to fire

Jean Marc Franssen & Thomas Gernay

December 2022

General Presentation of the software SAFIR®

1) Introduction to SAFIR
   a. Description of the software
   b. 3 steps procedure: fire, thermal, mechanical
   c. Selection of Finite Element type for thermal-mechanical analysis
   d. Description of the available FE
   e. General principle of a thermal-mechanical analysis with beams
   f. List of available materials
2) Introduction to the software environment
3) Examples, validation, and user community
4) Resources

Disclaimer: Under a license agreement between Gesval S.A. and the Johns Hopkins University, Dr. Gernay and the University are entitled to royalty distributions related to SAFIR. This arrangement has been reviewed and approved by the Johns Hopkins University in accordance with its conflict of interest policies.

1) Introduction to SAFIR
   a. **Description of the software**
   b. 3 steps procedure: fire, thermal, mechanical
   c. Selection of Finite Element type for thermal-mechanical analysis
   d. Description of the available FE
   e. General principle of a thermal-mechanical analysis with beams
   f. List of available materials
2) Introduction to the software environment
3) Examples, validation, and user community
4) Resources

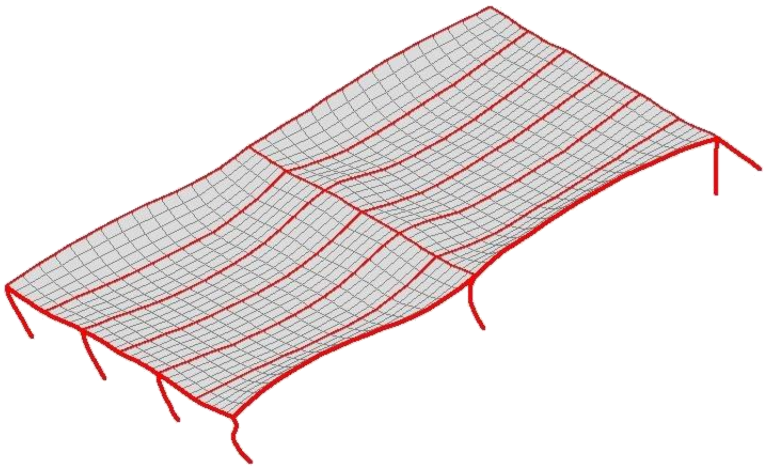
Description of SAFIR®

- 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.

Description of SAFIR®

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

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



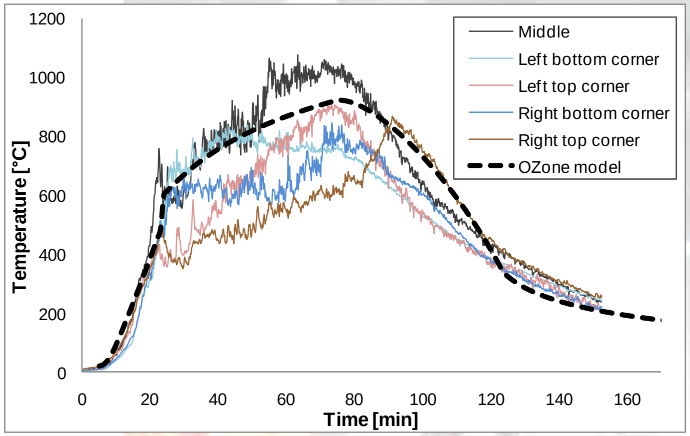
1) Introduction to SAFIR
   a. Description of the software
   b. 3 steps procedure: fire, thermal, mechanical
   c. Selection of Finite Element type for thermal-mechanical analysis
   d. Description of the available FE
   e. General principle of a thermal-mechanical analysis with beams
   f. List of available materials
2) Introduction to the software environment
3) Examples, validation, and user community
4) Resources

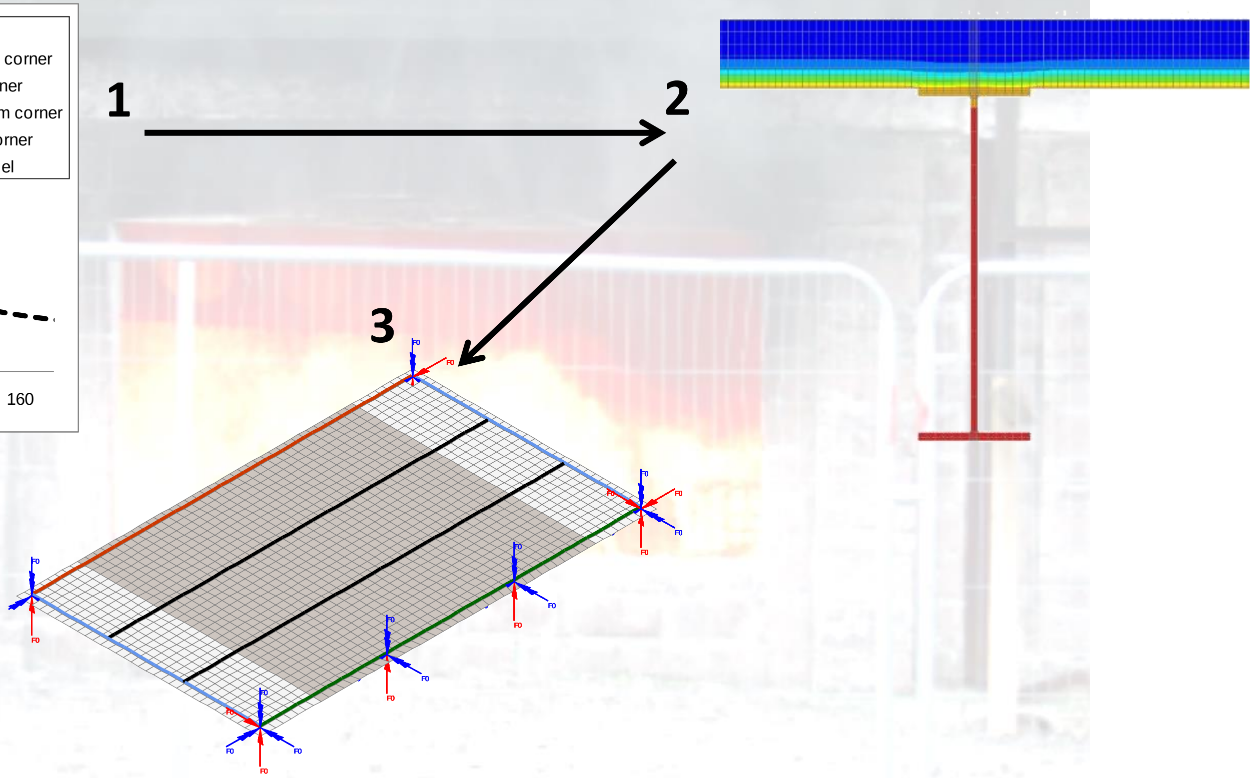
3 steps procedure in a SAFIR analysis

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 elongation as well as the reduction of strength and stiffness in the materials (mechanical analysis)

3 steps procedure in a SAFIR analysis

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

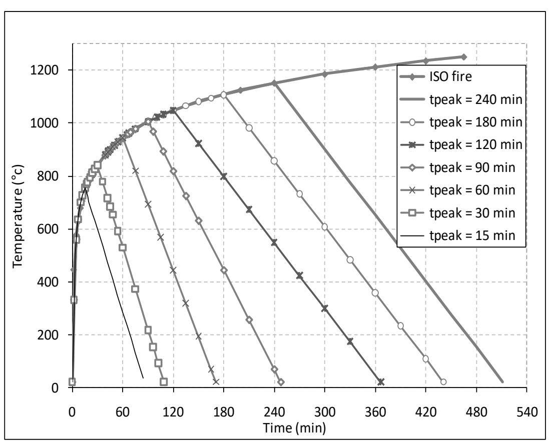


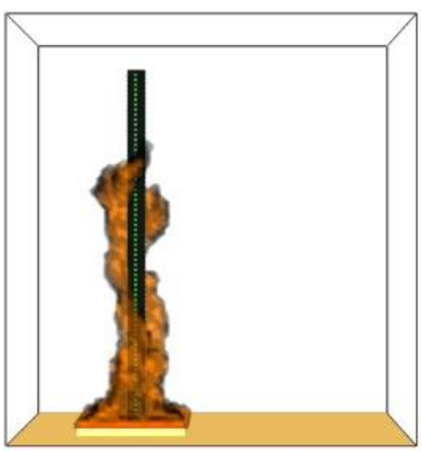


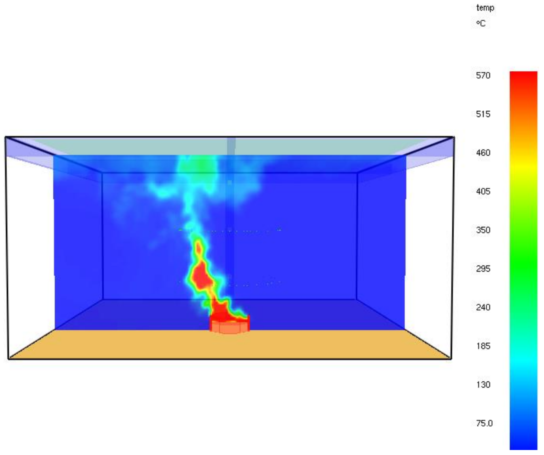
Vassart et al. (2012). "Large-scale fire test of unprotected cellular beam acting in membrane action". *Structures and Buildings, 165*(7), 327–334

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** (see "thermal" course):
  - 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)



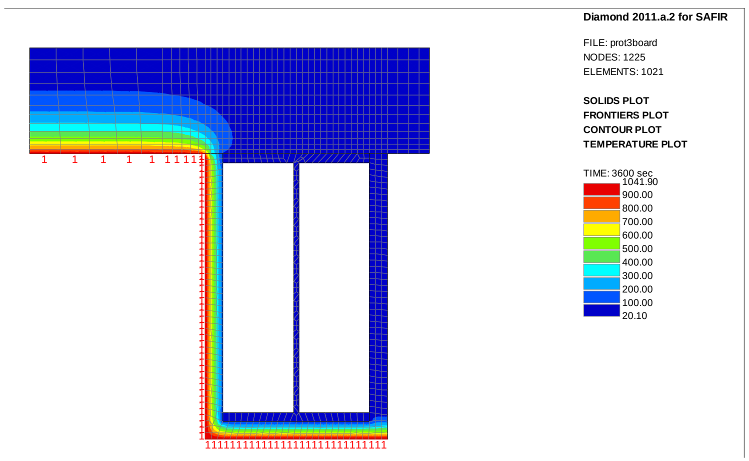


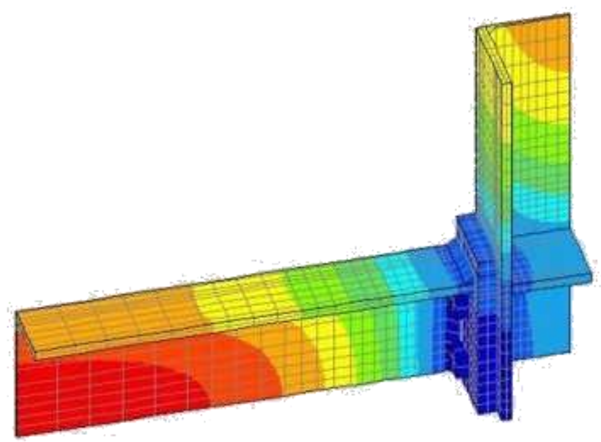


Step 2: Thermal analysis

SAFIR performs the transient thermal analysis to determine the **temperature distributions** in the structure

- 2D or 3D thermal calculations
- Finite elements: triangular, quadrangular, prismatic (6 or 8 nodes)
- Predefined thermal material models from Eurocodes: concrete, steel, wood, aluminum, gypsum
- User materials with user-defined temperature-dependent thermal properties





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 material models of Eurocodes : 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)

1) Introduction to SAFIR

   a. Description of the software
   b. 3 steps procedure: fire, thermal, mechanical
   c. **Selection of Finite Element type for thermal-mechanical analysis**
   d. Description of the available FE
   e. General principle of a thermal-mechanical analysis with beams
   f. List of available materials

2) Introduction to the software environment

3) Examples, validation, and user community

4) Resources

Types of Finite Elements

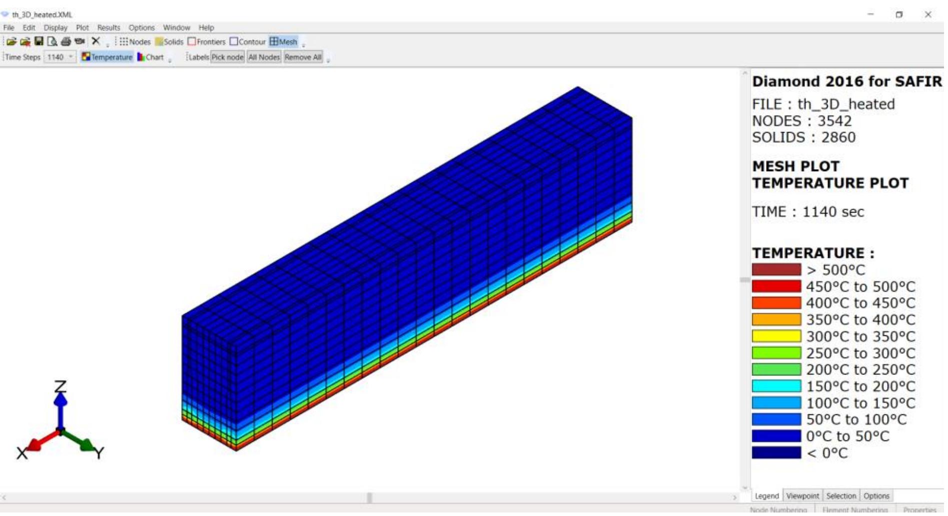
- Link between thermal and mechanical analyses

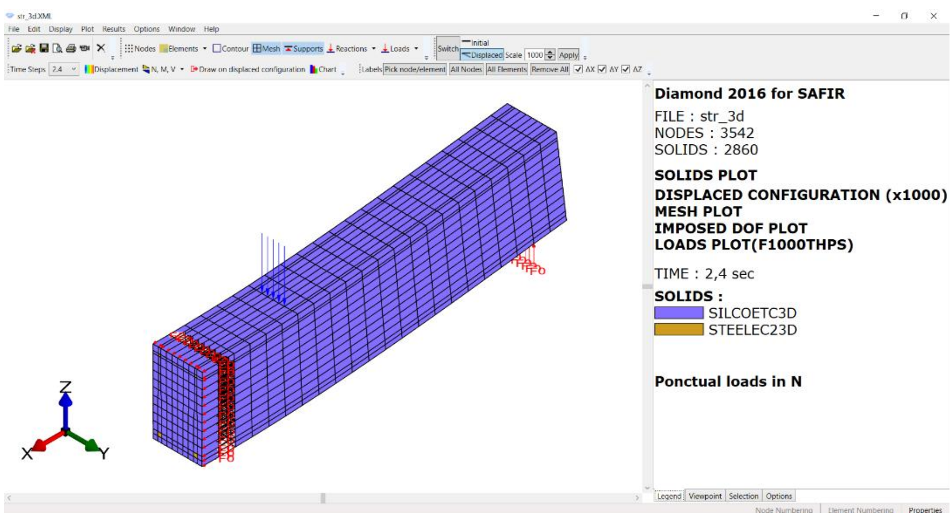
The type of model used for the thermal analysis depends on the type of model that will be used in the subsequent mechanical analysis

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)

Transfer of temperature information in mechanical finite elements

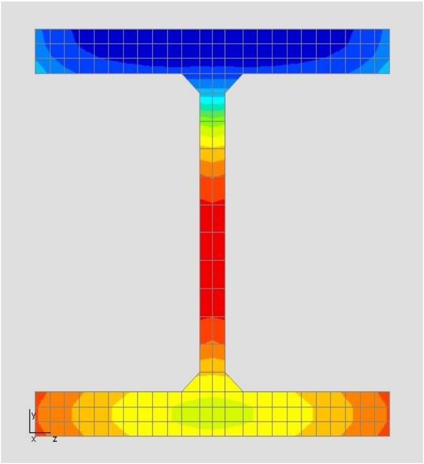
- For **3D solid elements**, the same discretization is used for the thermal and mechanical analyses so that the **temperatures are directly mapped** on the mechanical model.

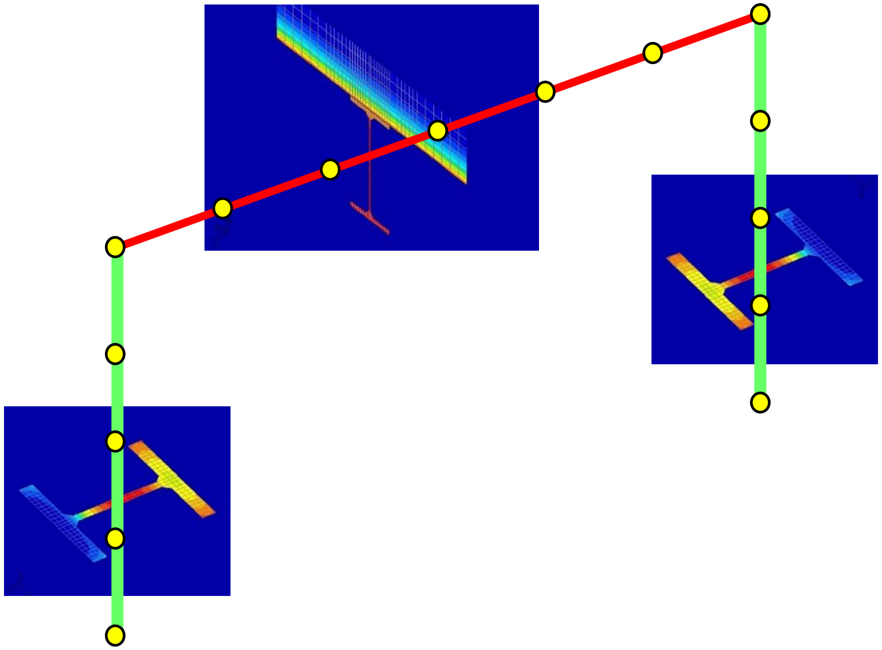




Transfer of temperature information in mechanical finite elements

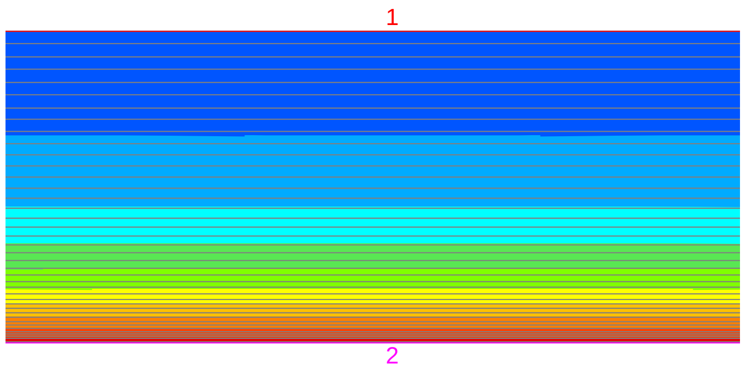
- For **beam elements**, the discretization of the section employed for the thermal analysis (calculation of the temperature at each node) is used in the form of **fibers** for the beam elements in the mechanical analysis. Thus, the determination of forces and stiffness in the section is based on the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

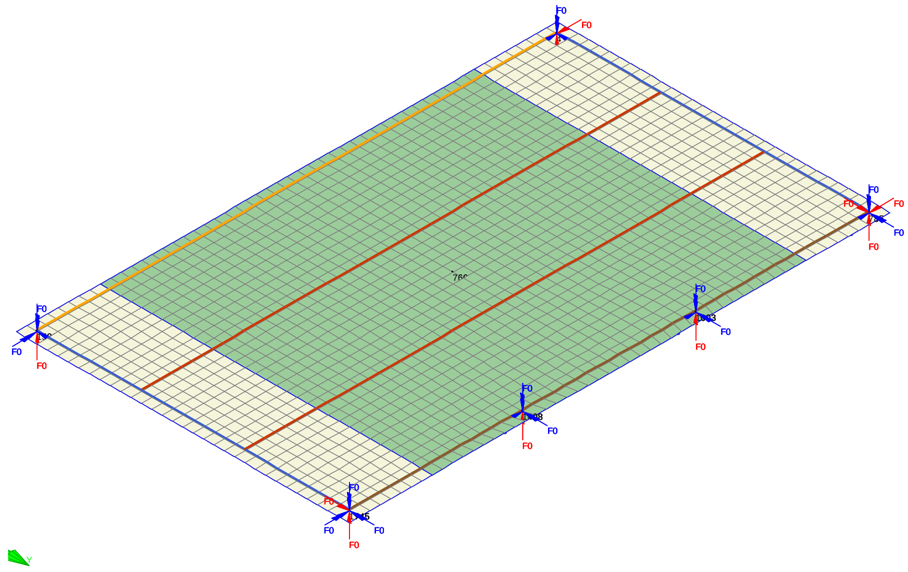




Transfer of temperature information in mechanical finite elements

- 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.





1) Introduction to SAFIR

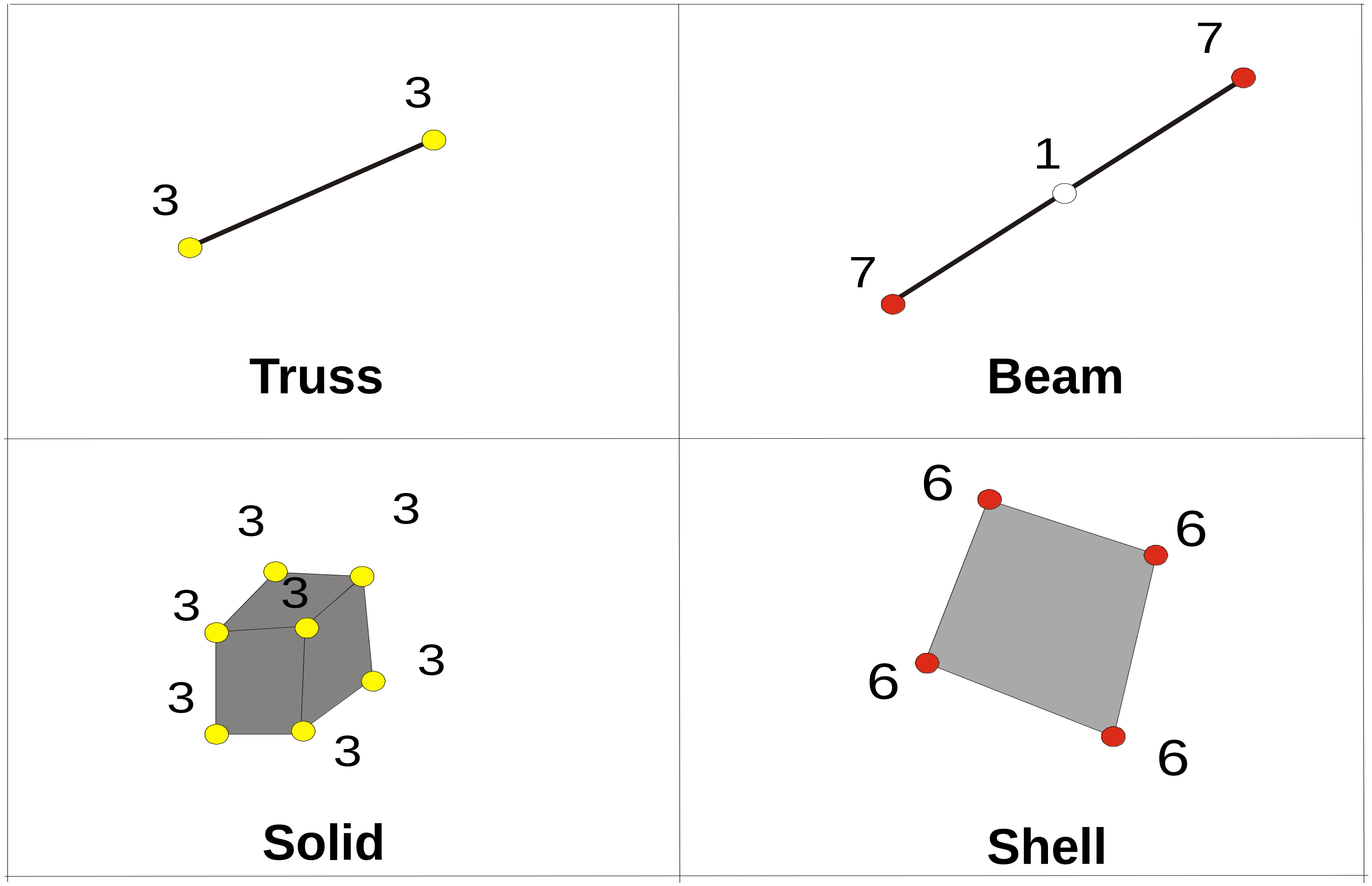
   a. Description of the software
   b. 3 steps procedure: fire, thermal, mechanical
   c. Selection of Finite Element type for thermal-mechanical analysis
   d. **Description of the available FE**
   e. General principle of a thermal-mechanical analysis with beams
   f. List of available materials

2) Introduction to the software environment

3) Examples, validation, and user community

4) Resources

Discretization for mechanical analysis



Truss FE

- One single point of integration (i.e. one material, temperature and stress level) for each element
- 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

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)

Shell FE

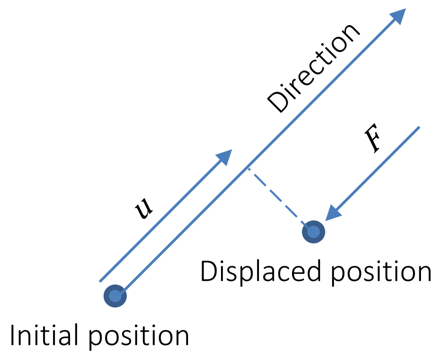
- Quadrangle based on 4 nodes
- 6 DoF at each node: 3 translations and 3 rotations
- Integration on the plane is performed numerically using 4 points of Gauss
- Integration on the thickness is performed numerically with the user choosing the number of Gauss points (from 2 if membrane behavior dominates 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)

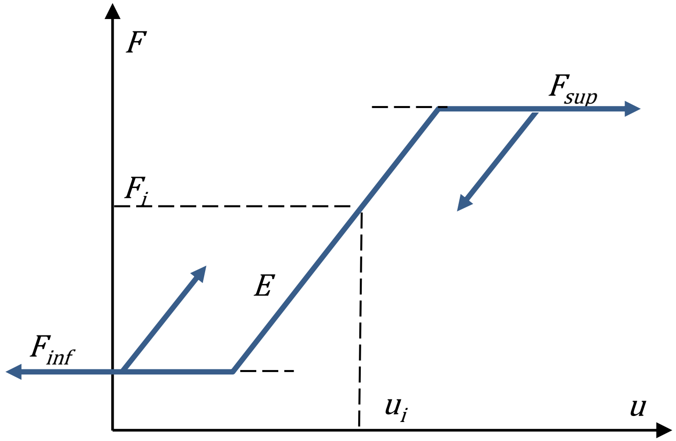
Solid FE

- 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

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





1) Introduction to SAFIR

a. Description of the software

b. 3 steps procedure: fire, thermal, mechanical

c. Selection of Finite Element type for thermal-mechanical analysis

d. Description of the available FE

e. General principle of a thermal-mechanical analysis with beams

f. List of available materials

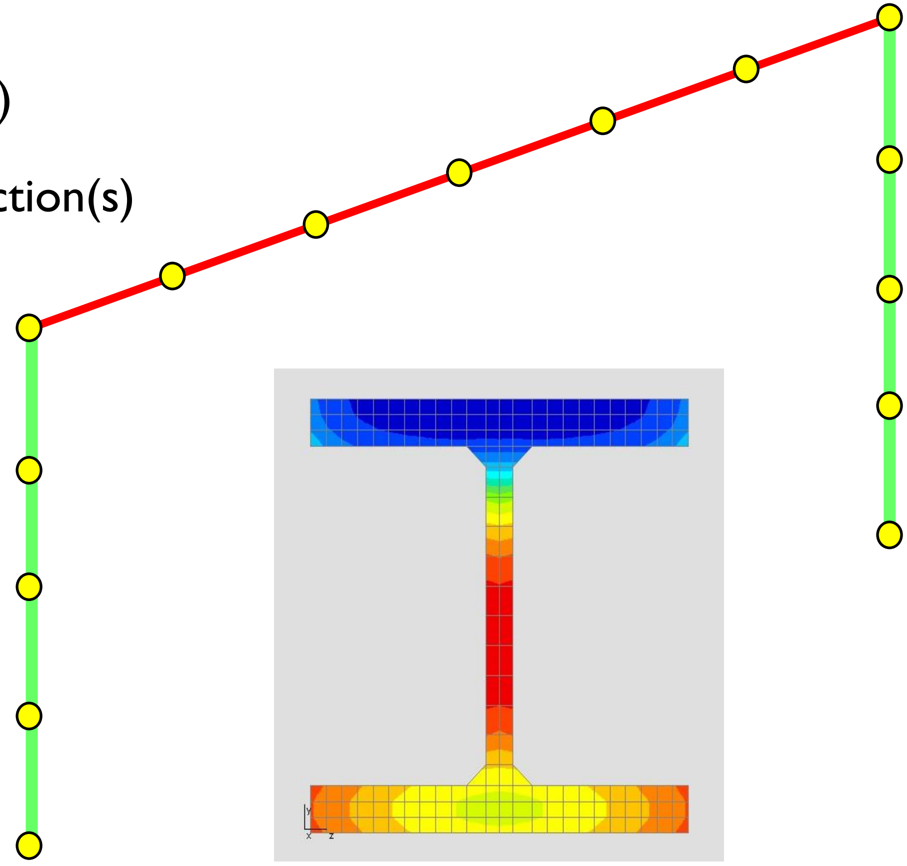
2) Introduction to the software environment

3) Examples, validation, and user community

4) Resources

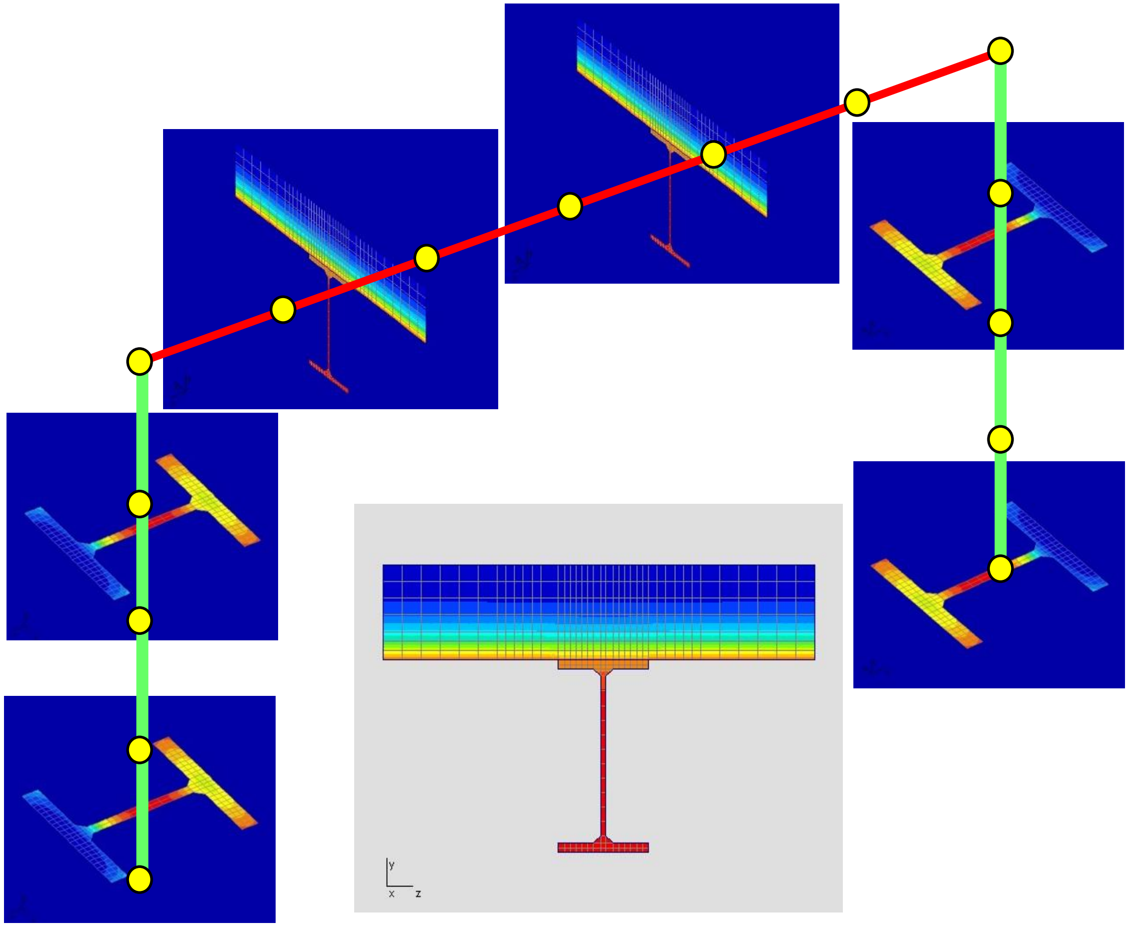
General principle of a mechanical analysis based on beam FE (1/2)

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)



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



1) Introduction to SAFIR
   a. Description of the software
   b. 3 steps procedure: fire, thermal, mechanical
   c. Selection of Finite Element type for thermal-mechanical analysis
   d. Description of the available FE
   e. General principle of a thermal-mechanical analysis with beams
   f. List of available materials
2) Introduction to the software environment
3) Examples, validation, and user community
4) Resources

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	X	
User_Conc			X		

1) Introduction to SAFIR
   a. Description of the software
   b. 3 steps procedure: fire, thermal, mechanical
   c. Selection of Finite Element type for thermal-mechanical analysis
   d. Description of the available FE
   e. General principle of a thermal-mechanical analysis with beams
   f. List of available materials

2) **Introduction to the software environment**

3) Examples, validation, and user community

4) Resources

Preprocessor GmSAFIR

- GmSAFIR is a GUI preprocessor
- Allows the generation of input files for 2D / 3D, thermal / torsional / structural models
- GmSAFIR is free and open source, see: https://github.com/gmsafir/gmsafir

Note: use of GmSAFIR is not strictly necessary – SAFIR reads ASCII input files which can be generated by text editor, user-developed code, etc.

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 for free at: https://www.uee.uliege.be/cms/c_4016387/fr/ueenew-ressources-sur-safir

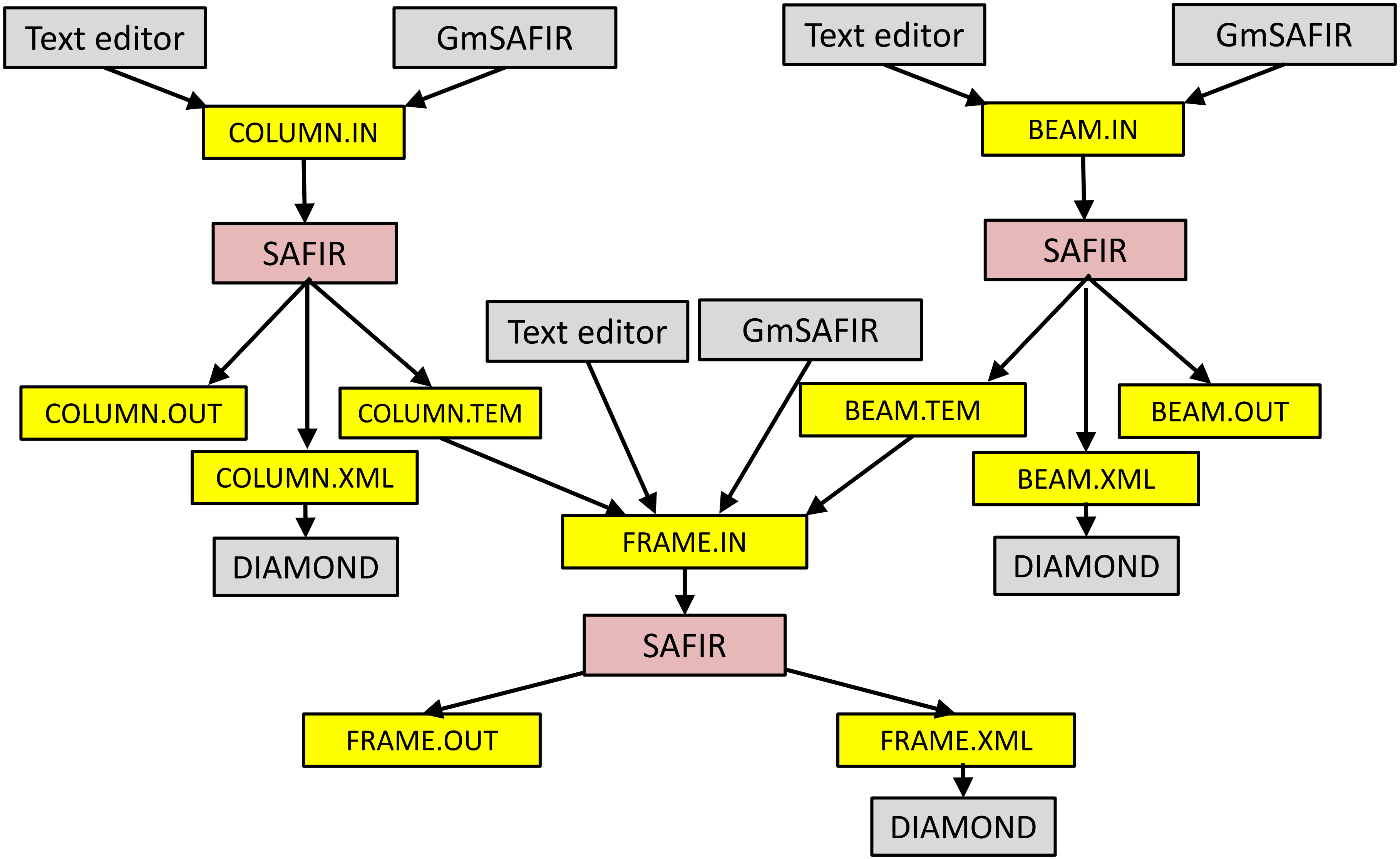
Organization of the files for a typical calculation – Example for:

- 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.


Text editor
GmSAFIR
COLUMN.IN
SAFIR
COLUMN.OUT
COLUMN.XML
COLUMN.TEM
DIAMOND
Text editor
GmSAFIR
BEAM.IN
SAFIR
BEAM.TEM
BEAM.XML
BEAM.OUT
DIAMOND
Text editor
GmSAFIR
FRAME.IN
SAFIR
FRAME.OUT
FRAME.XML
DIAMOND

1) Introduction to SAFIR
   a. Description of the software
   b. 3 steps procedure: fire, thermal, mechanical
   c. Selection of Finite Element type for thermal-mechanical analysis
   d. Description of the available FE
   e. General principle of a thermal-mechanical analysis with beams
   f. List of available materials

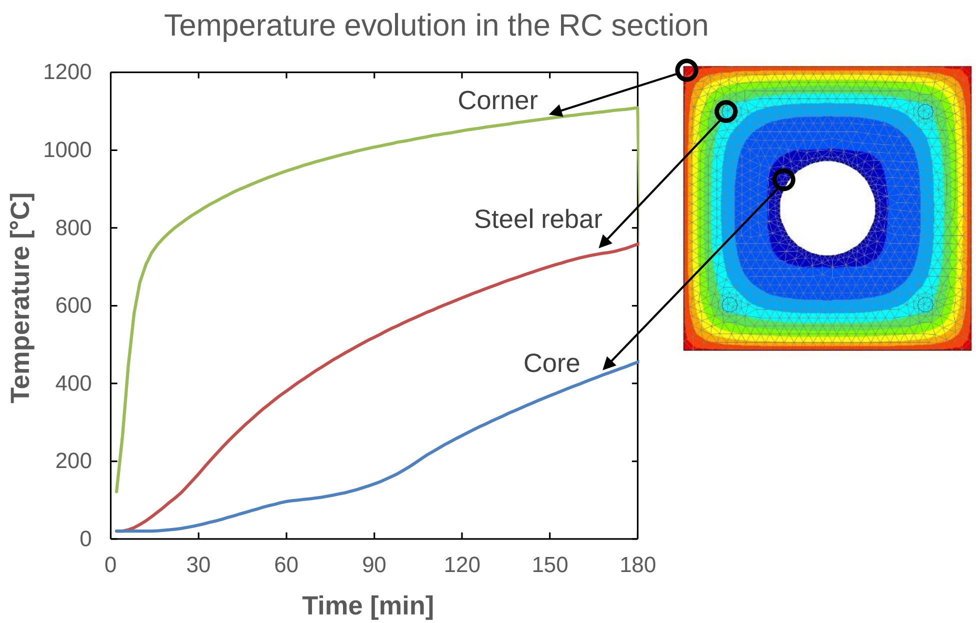
2) Introduction to the software environment

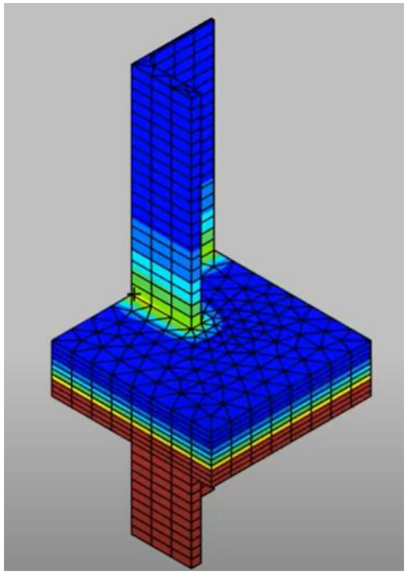
3) Examples, validation, and user community

4) Resources

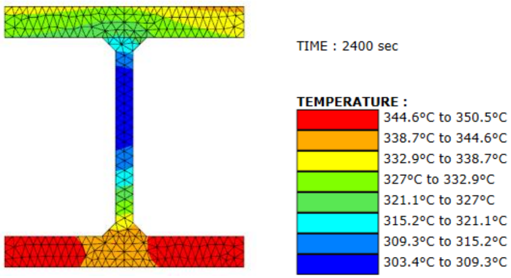
Examples of thermal analysis

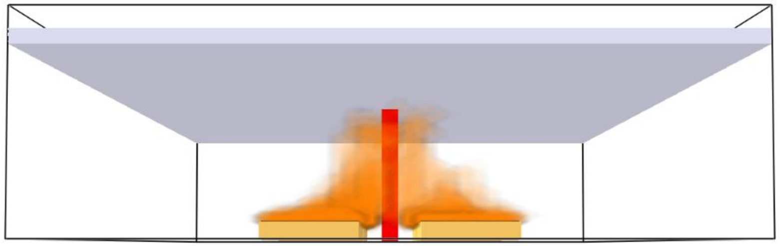
2D thermal calculation - Reinforced concrete column with hollow core
1097 nodes - 2012 triangular elements





3D thermal calculation
H-section through a slab

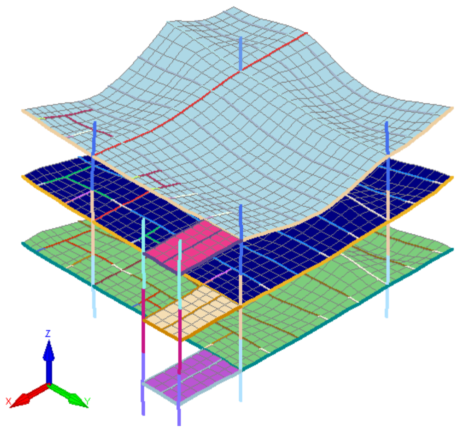


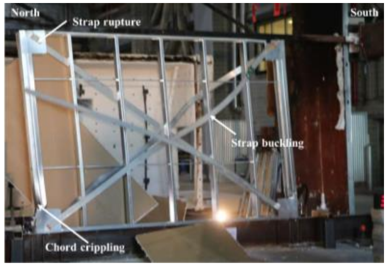


2D thermal calculation – Steel section
Interfacing with FDS simulation (localized fire)

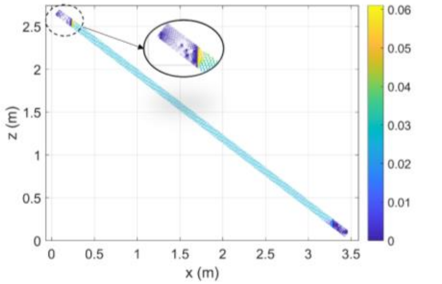
Example of mechanical analysis

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

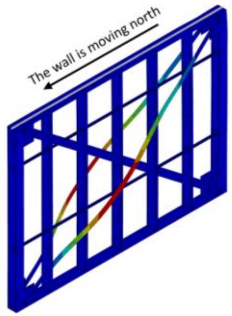




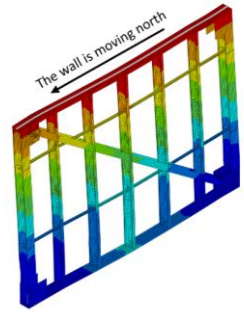
(a) Failure by rupture of the tension strap [1]



(b) Maximum principal strains in the tensile strap obtained by numerical modeling



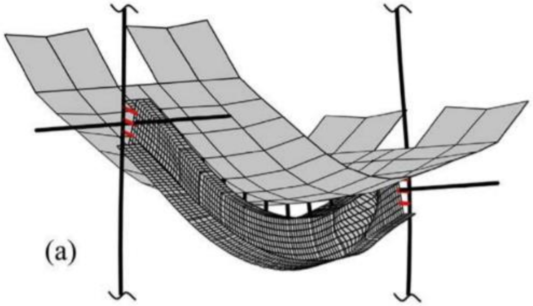
(c) Buckling of the compression strap in the numerical model (displaced scale: 10)



(d) Displacements at failure in the numerical model (displaced scale: 1)

Drury, M. M., Kordosky, A. N., & Quiel, S. E. (2020). Structural fire resistance of partially restrained, partially composite floor beams, II: Modeling. Journal of Constructional Steel Research, 167, 105946. https://doi.org/10.1016/j.jcsr.2020.105946

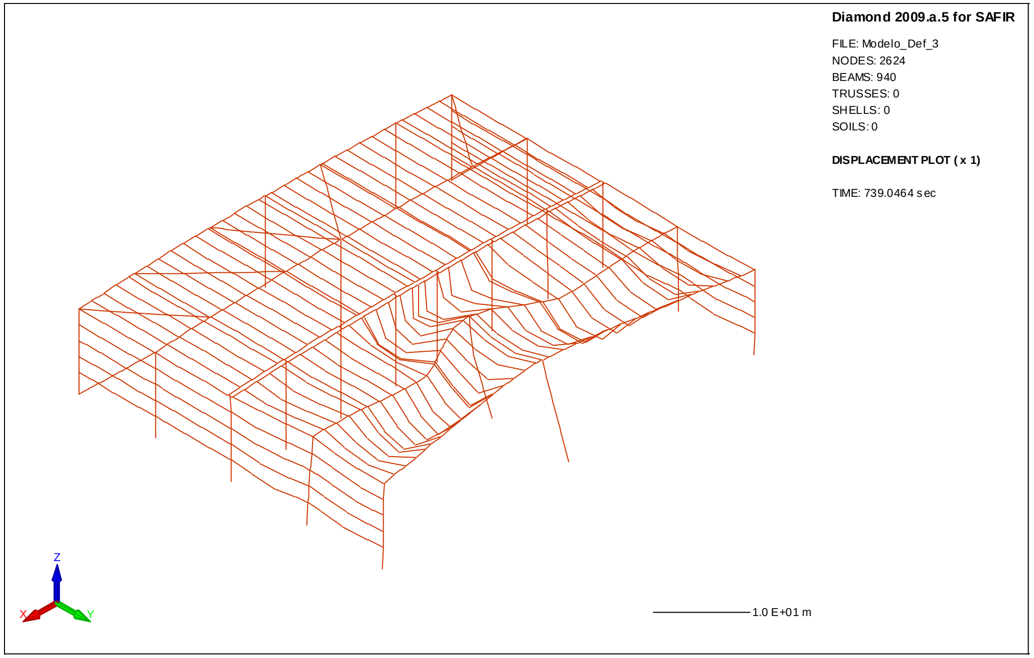
Ni, S., Yan, X., Hoehler, M. & Gernay, T. (2022). Numerical modeling of the post-fire performance of strap-braced cold-formed steel shear walls, *Thin-Walled Structures, 171* https://doi.org/10.1016/j.tws.2021.108733



Example of mechanical analysis of a full scale test

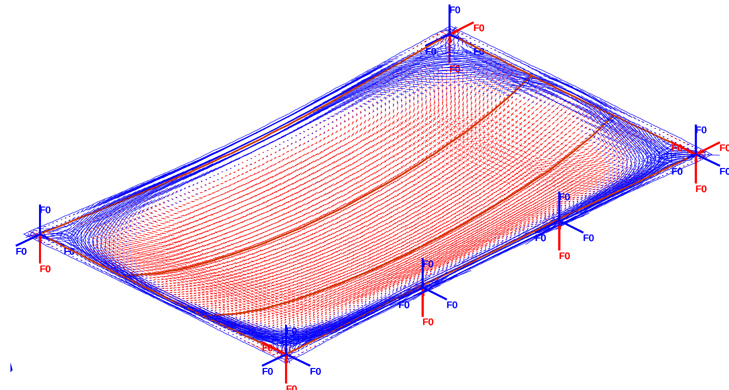
3D mechanical calculation – Steel frame

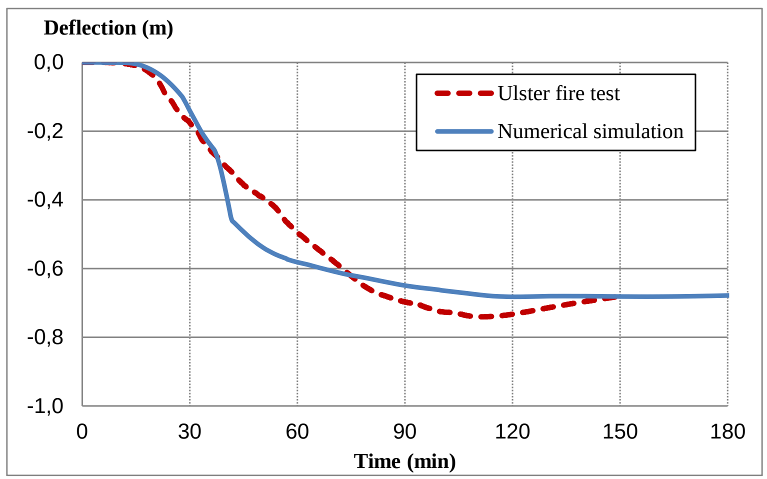
Flumilog test, INERIS, France
2 624 nodes, 940 beam elements



Example of mechanical analysis of a full scale test

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

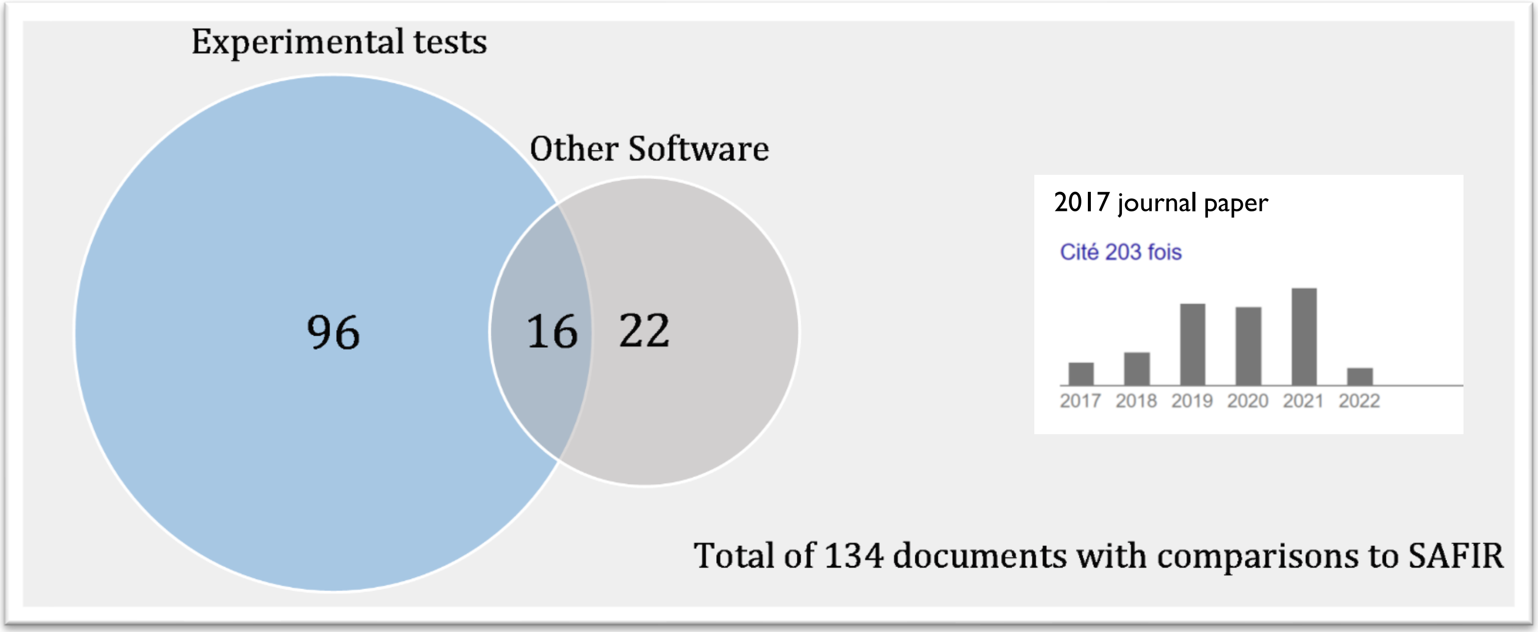




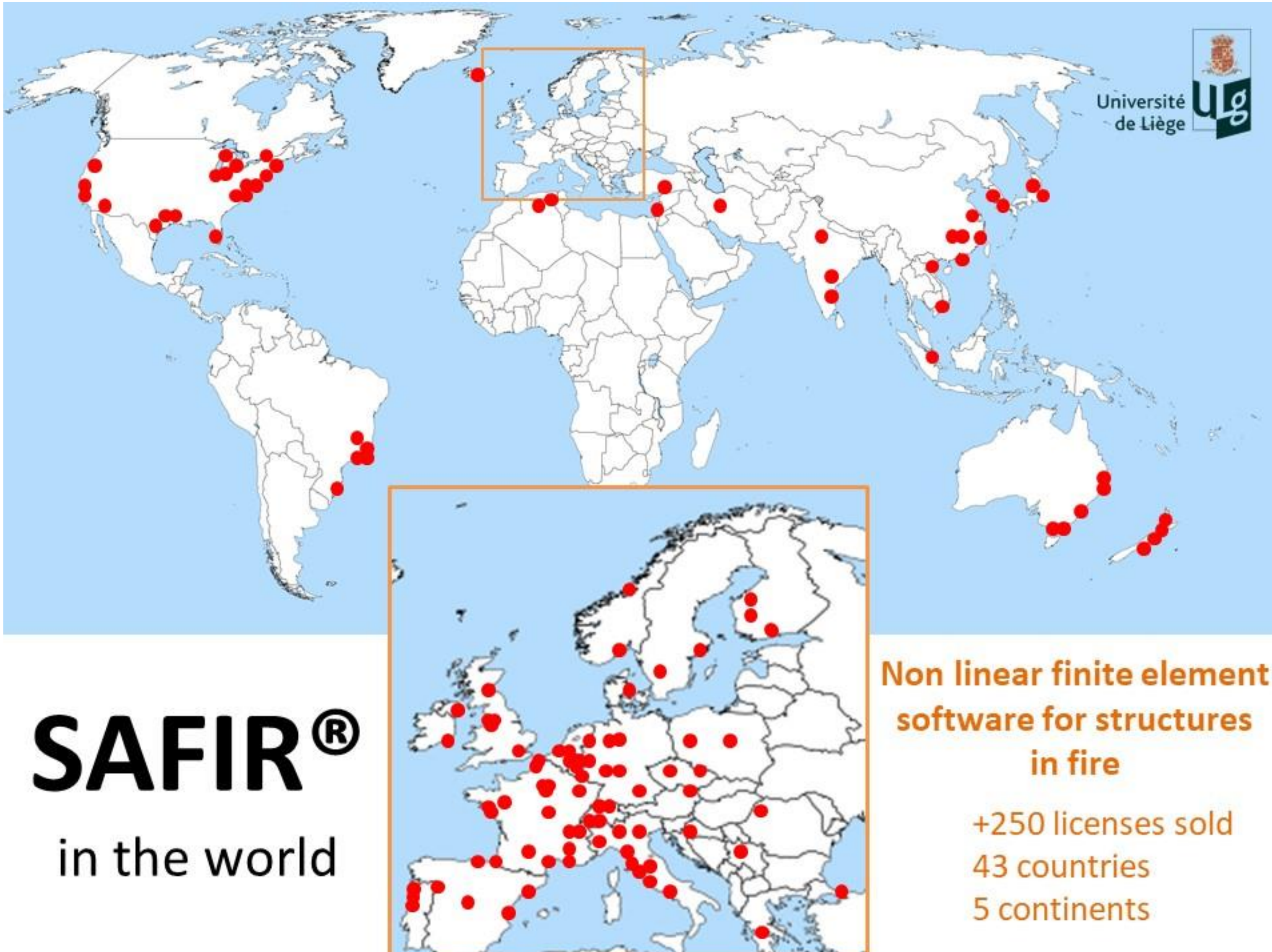
Vassart et al. (2012). *Structures and Buildings, 165*(7), 327–334

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.
- 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 https://www.uee.uliege.be/cms/c_4016387/fr/ueenew-ressources-sur-safir

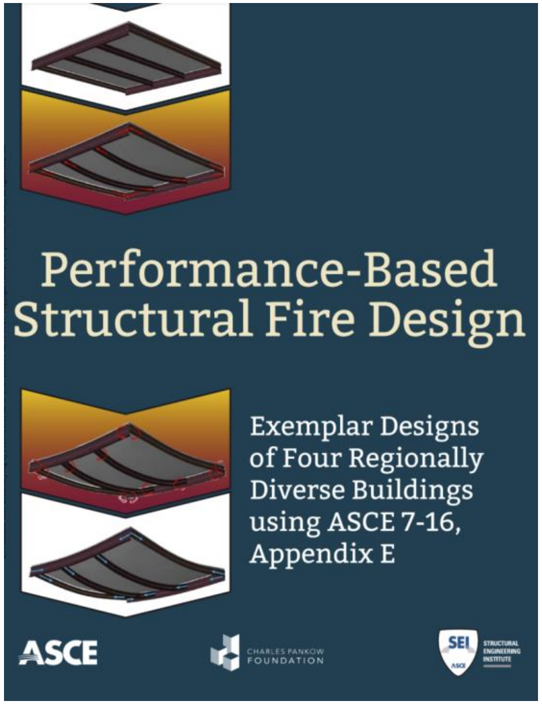


Users


Université de Liège
SAFIR®
in the world
Non linear finite element software for structures in fire
+250 licenses sold
43 countries
5 continents

Example of published application: ASCE-CPF Study

SGH, Walter P Moore, Thornton Tomasetti



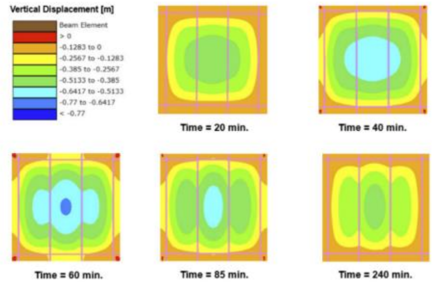


Figure 7-24. Floor slab deflection (Case 10) (interior bay fire).
Source: Courtesy of Simpson Gumpertz & Heger (2019), and SAFIR Software ©2019.

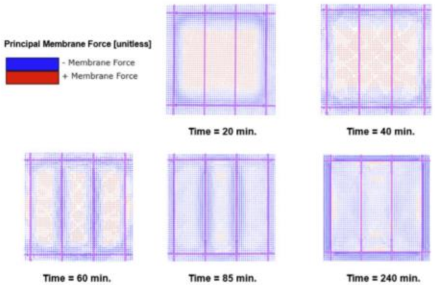


Figure 7-25. Floor slab principal membrane forces (Case 10) (interior bay fire).
Source: Courtesy of Simpson Gumpertz & Heger (2019), and SAFIR Software ©2019.

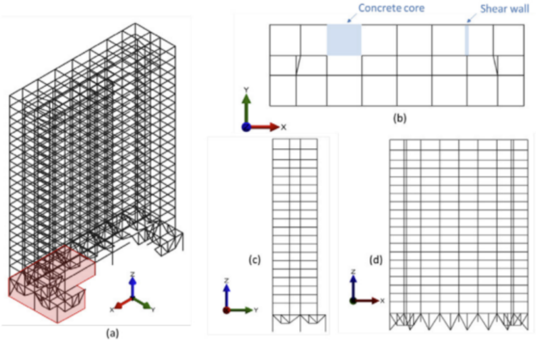


Figure 9-19. Building geometry in SAFIR: (a) isoparametric view, (b) plan XY view of the 20 residential floors with concrete core and shear wall locations, (c) side YZ view, and (d) side XZ view.
Source: Courtesy of Thornton Tomasetti (2019).

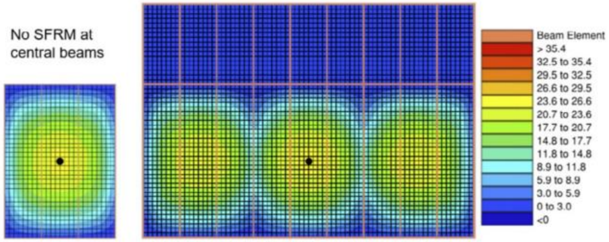


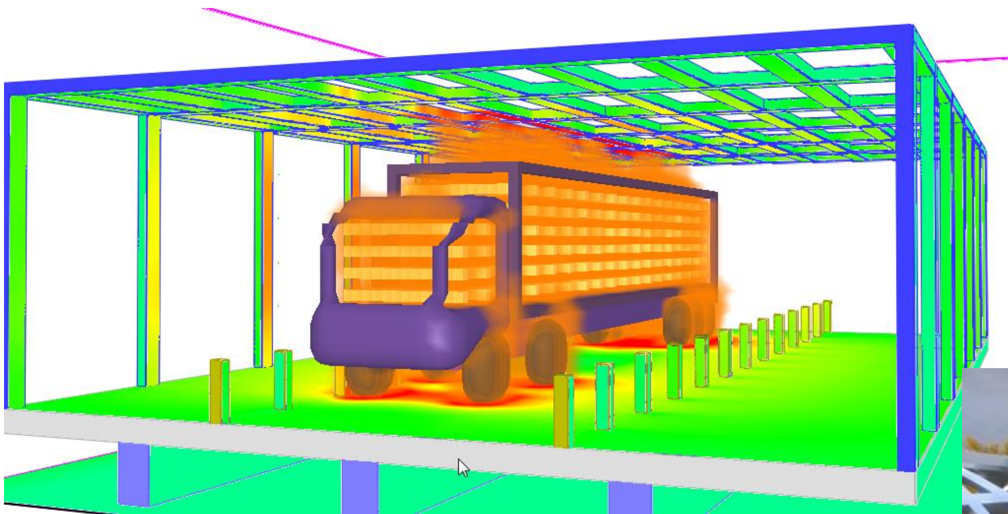
Figure 10-10. SAFIR single- and multibay model displacement contour at 20th min for Fire Case 1.
Source: Courtesy of Walter P Moore (2019).
Note: Legend is in inches.

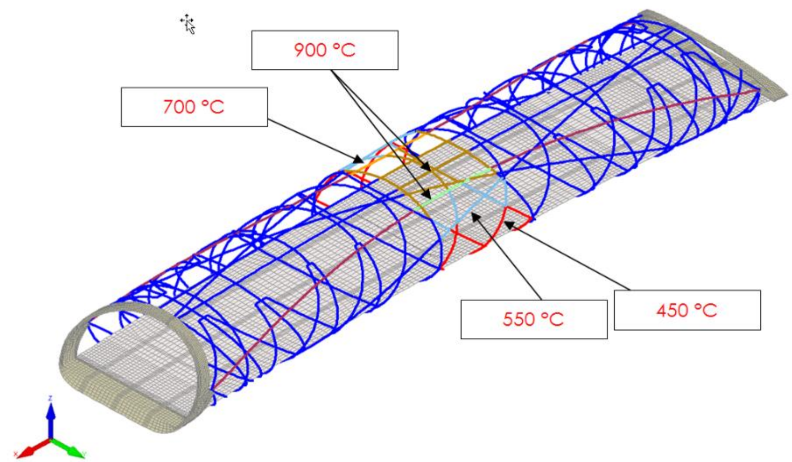
Prepared by the Structural Engineering Institute (SEI)
of the American Society of Civil Engineers (ASCE)

https://ascelibrary.org/doi/book/10.1061/9780784482698

Example of published commercial application

MP Ingénieurs Conseils, Switzerland – Wilsdorf bridge in Geneva

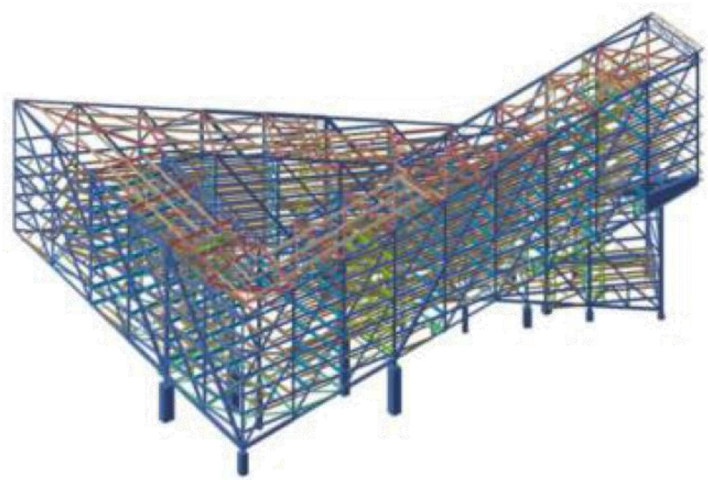




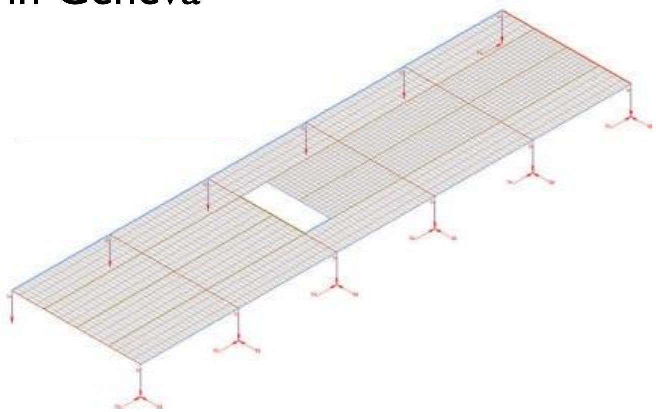
Tonicello, E. et al. (2012). *Proc. 7th Structures in Fire Conf.*

Example of published commercial application

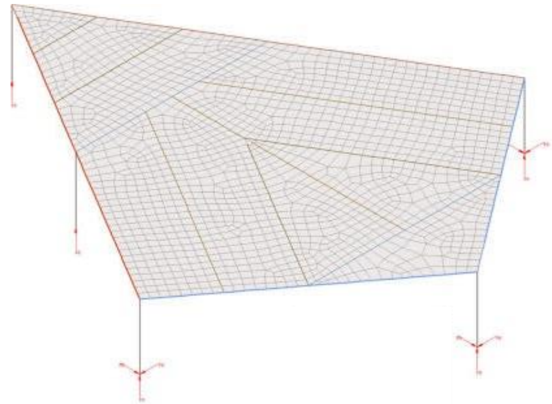
INGENI, Switzerland – Japan Tobacco Headquarter in Geneva



Model for ambient temperature analysis (not SAFIR)



SAFIR models for fire analysis

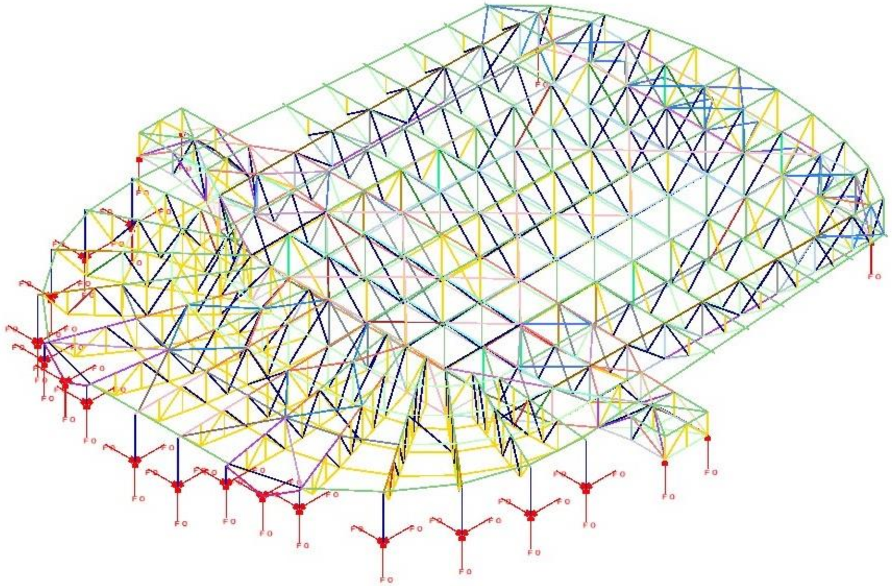


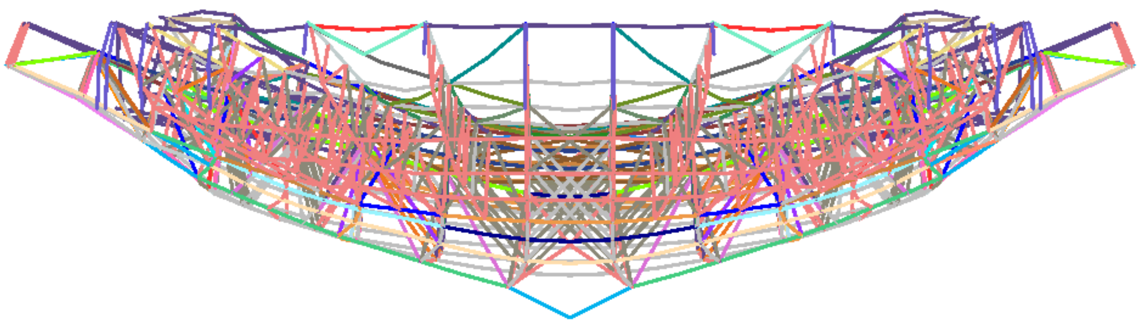
Lelli, L., Loutan, J. (2017). *Journal of Structural Fire Engineering*.

Example of published commercial application

BuroHappold Engineering, UK

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





1) Introduction to SAFIR
   a. Description of the software
   b. 3 steps procedure: fire, thermal, mechanical
   c. Selection of Finite Element type for thermal-mechanical analysis
   d. Description of the available FE
   e. General principle of a thermal-mechanical analysis with beams
   f. List of available materials

2) Introduction to the software environment

3) Examples, validation, and user community

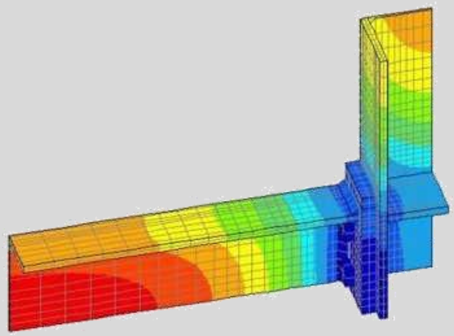
4) Resources

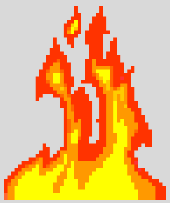
Resources

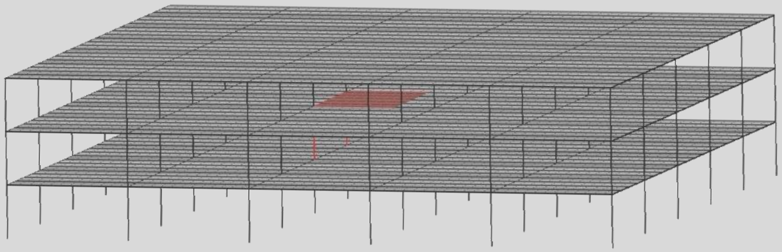
- Manuals, worked examples and input files can be downloaded for free
- Manuals: https://www.uee.uliege.be/cms/c_4016387/fr/ueenew-ressources-sur-safir
- Examples: https://www.uee.uliege.be/cms/c_4016388/fr/ueenew-safir-application-examples
- More examples: https://mars.jhu.edu/safir/worked-examples/
- Video tutorials: https://www.youtube.com/channel/UC0E-fNxuxk0pQORHy89Lw6g
- GmSAFIR: https://github.com/gmsafir/gmsafir/tree/master
- Contact us at safir@uliege.be
- Training sessions can be organized on demand

Reference

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







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.

Disclaimer: Under a license agreement between Gesval S.A. and the Johns Hopkins University, Dr. Gernay and the University are entitled to royalty distributions related to SAFIR. This arrangement has been reviewed and approved by the Johns Hopkins University in accordance with its conflict of interest policies.