SAFIR®
A software for modeling the behavior of structures subjected to the fire

Course by

Jean Marc Franssen & Thomas Gernay

LIÈGE université

JOHNS HOPKINS WHITING SCHOOL of ENGINEERING

Last revision: 28/01/2020

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 and validation
4) User community and references
5) Resources

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 and validation
4) User community and references
5) Resources

Description of SAFIR®

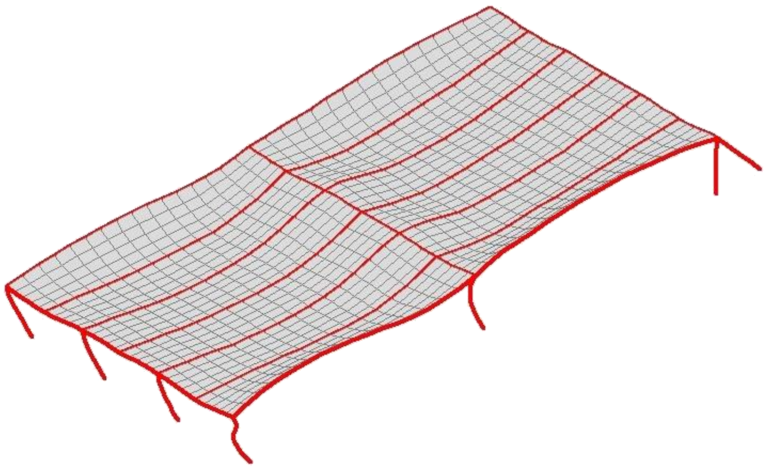
SAFIR is a **computer program** that models the **behavior of building structures subjected to fire**. The structure can be made of a 3D skeleton of linear elements such as **beams and columns**, in conjunction with planar elements such as **slabs and walls**. Volumetric elements can be used for analysis 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 non linear **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*)



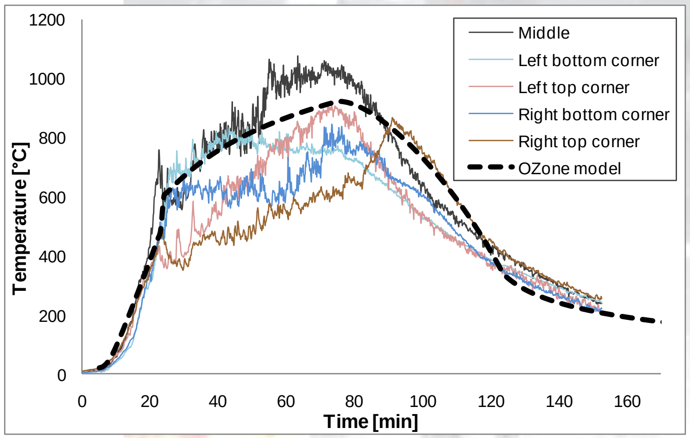
1) Introduction to SAFIR
    a. Description of the software
    b. 3 steps procedure: fire, thermal, mechanical
    c. Selection of Finite Element type for thermal-mechanical analysis
    d. Description of the available FE
    e. General principle of a thermal-mechanical analysis with beams
    f. List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

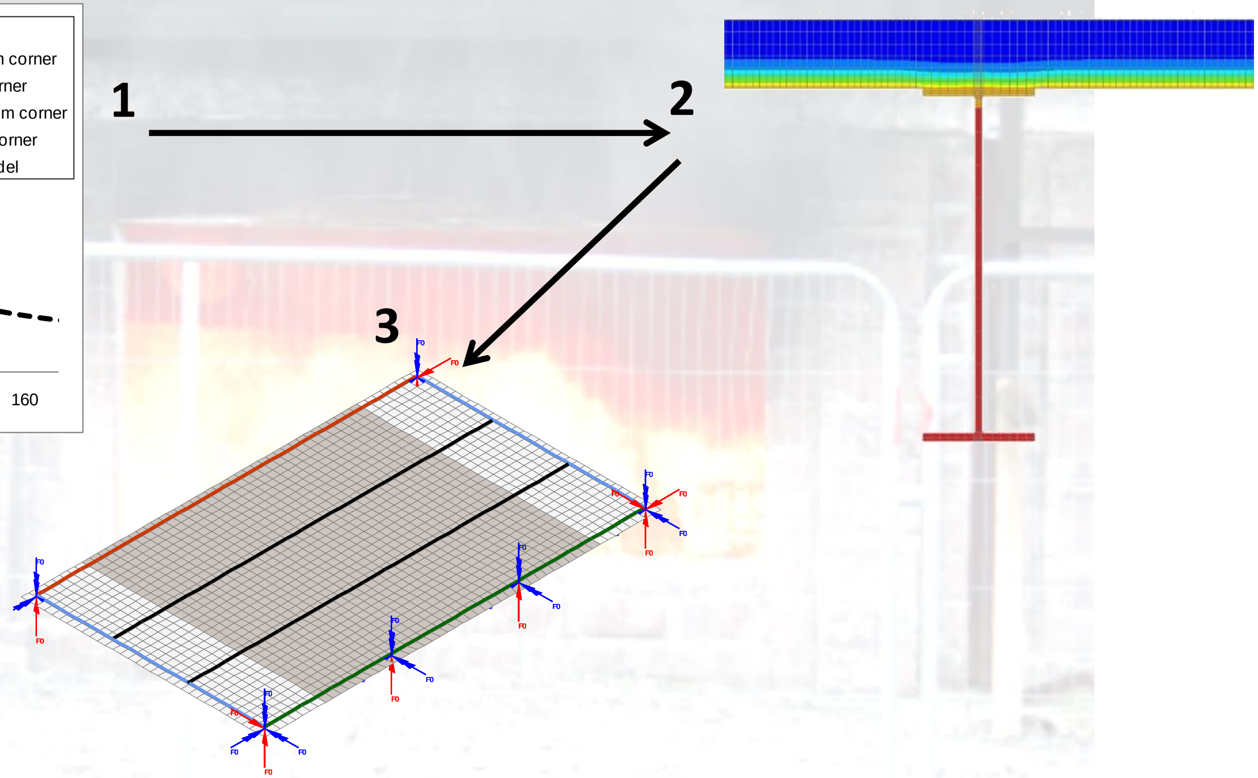
3 steps procedure in a SAFIR analysis

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 elongations 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

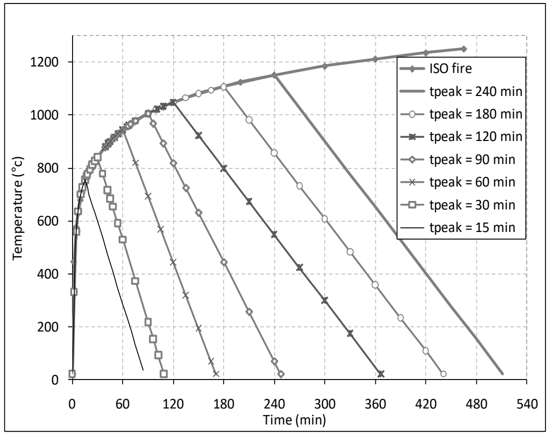


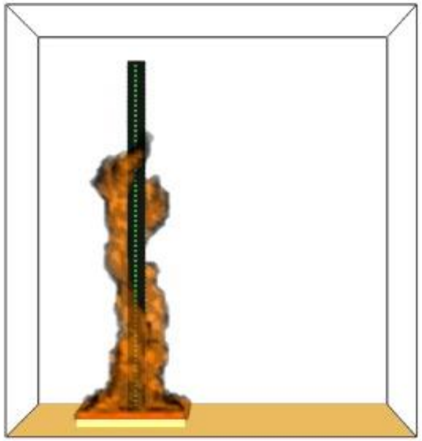


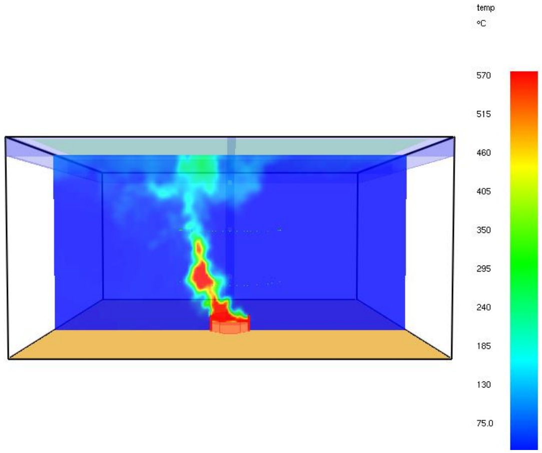
Vassart et al. (2012). “Large-scale fire test of unprotected cellular beam acting in membrane action“. *Structures and Buildings, 165*(7), 327–334

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)



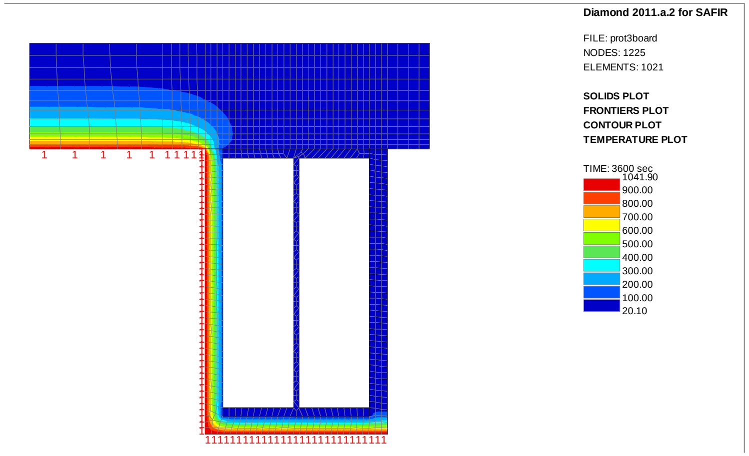


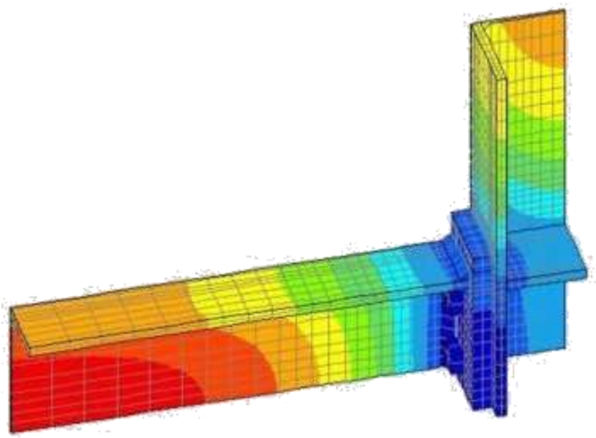


Step 2: Thermal analysis

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 and validation

4) User community and references

5) Resources

Types of Finite Elements

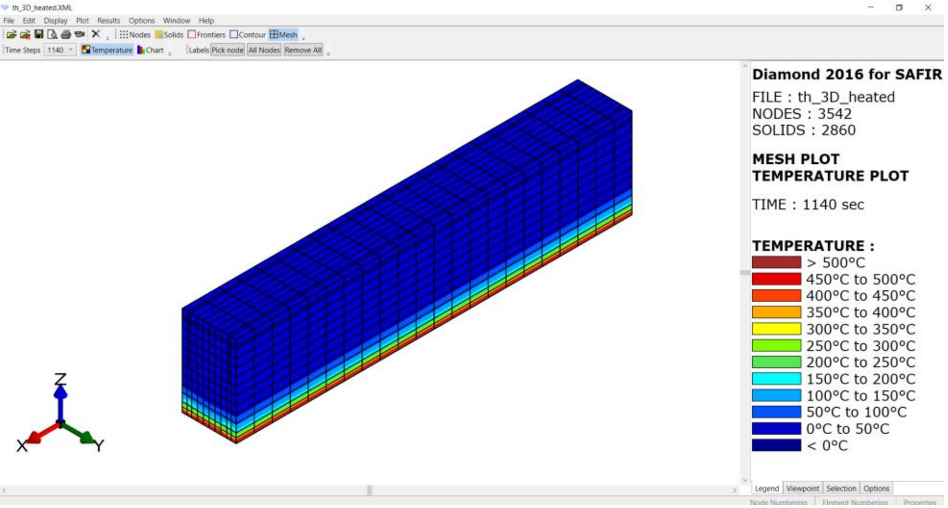
- Link between thermal and mechanical analyses

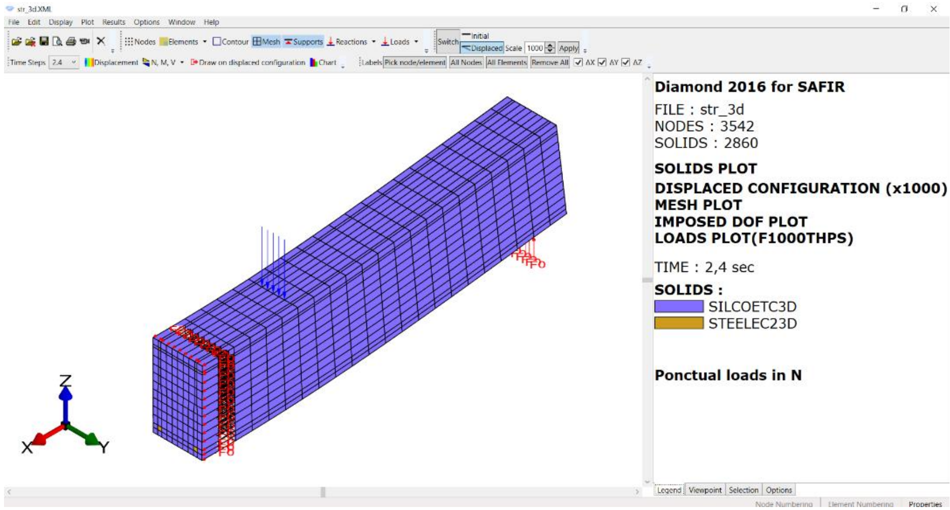
The type of model used for the thermal analysis depends on the type of model that will be used in the subsequent mechanical analysis

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

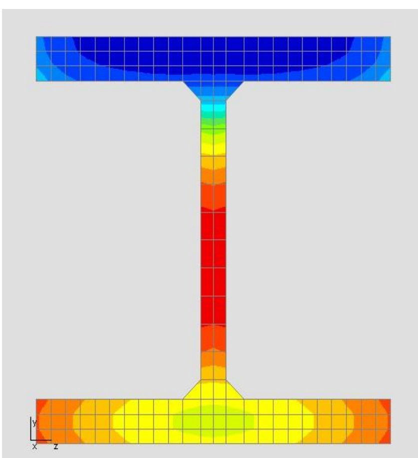
- For **3D solid elements**, the same discretization is used for the thermal and mechanical analyses so that the **temperatures are directly mapped** on the mechanical model.

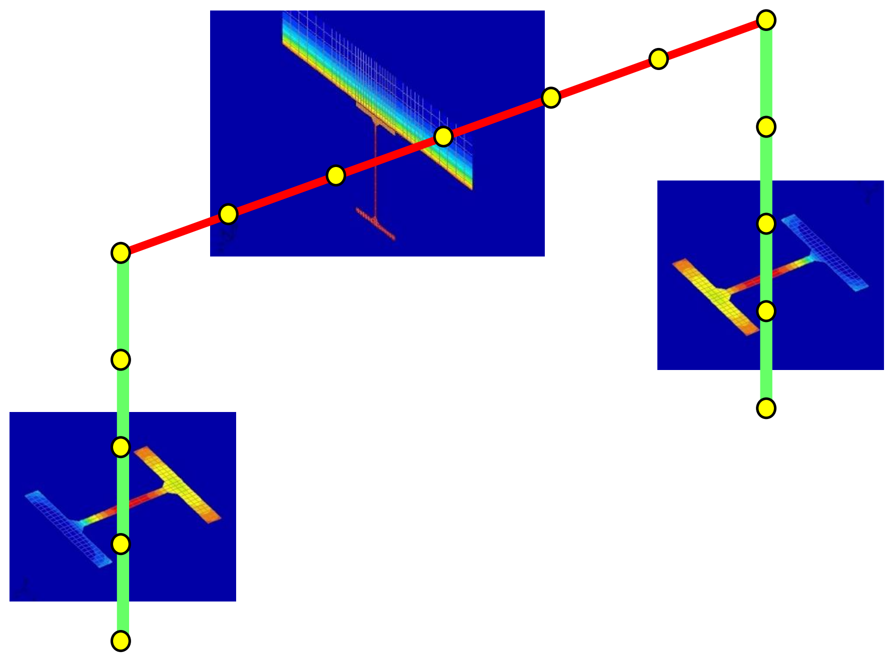




Transfer of temperature information in mechanical finite elements

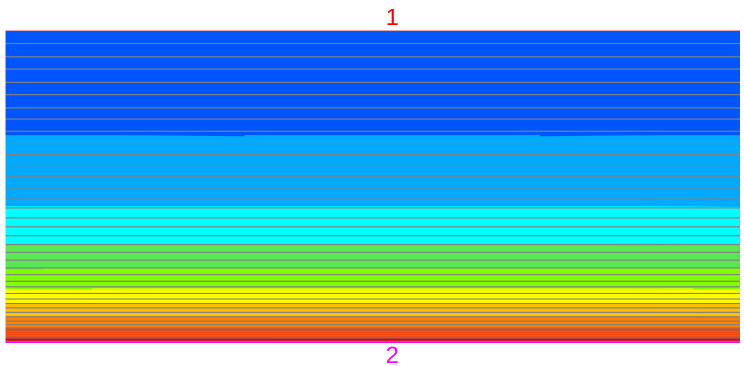
- For **beam elements**, the discretization of the section employed for the thermal analysis (calculation of the temperature at each node) is used in the form of **fibers** for the beam elements in the mechanical analysis. Thus, the determination of forces and stiffness in the section is based on the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

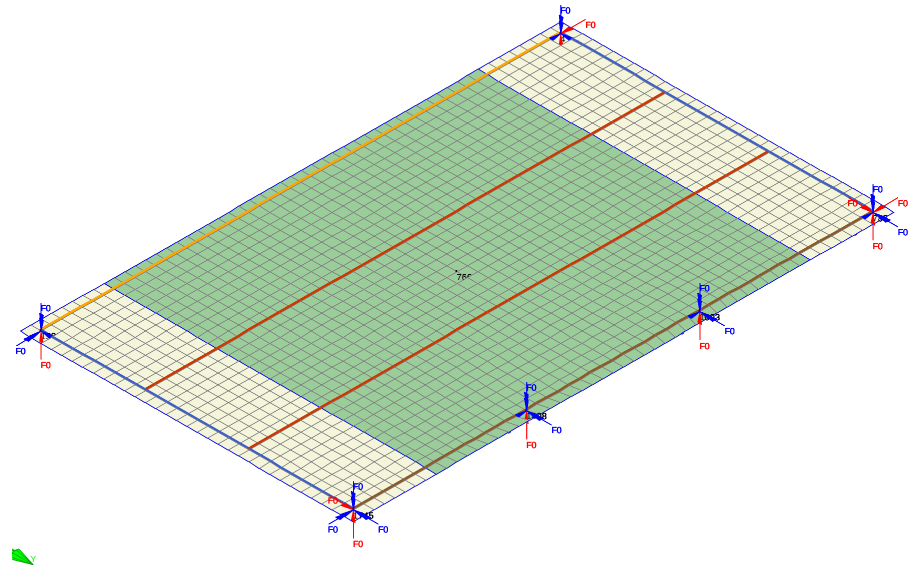




Transfer of temperature information in mechanical finite elements

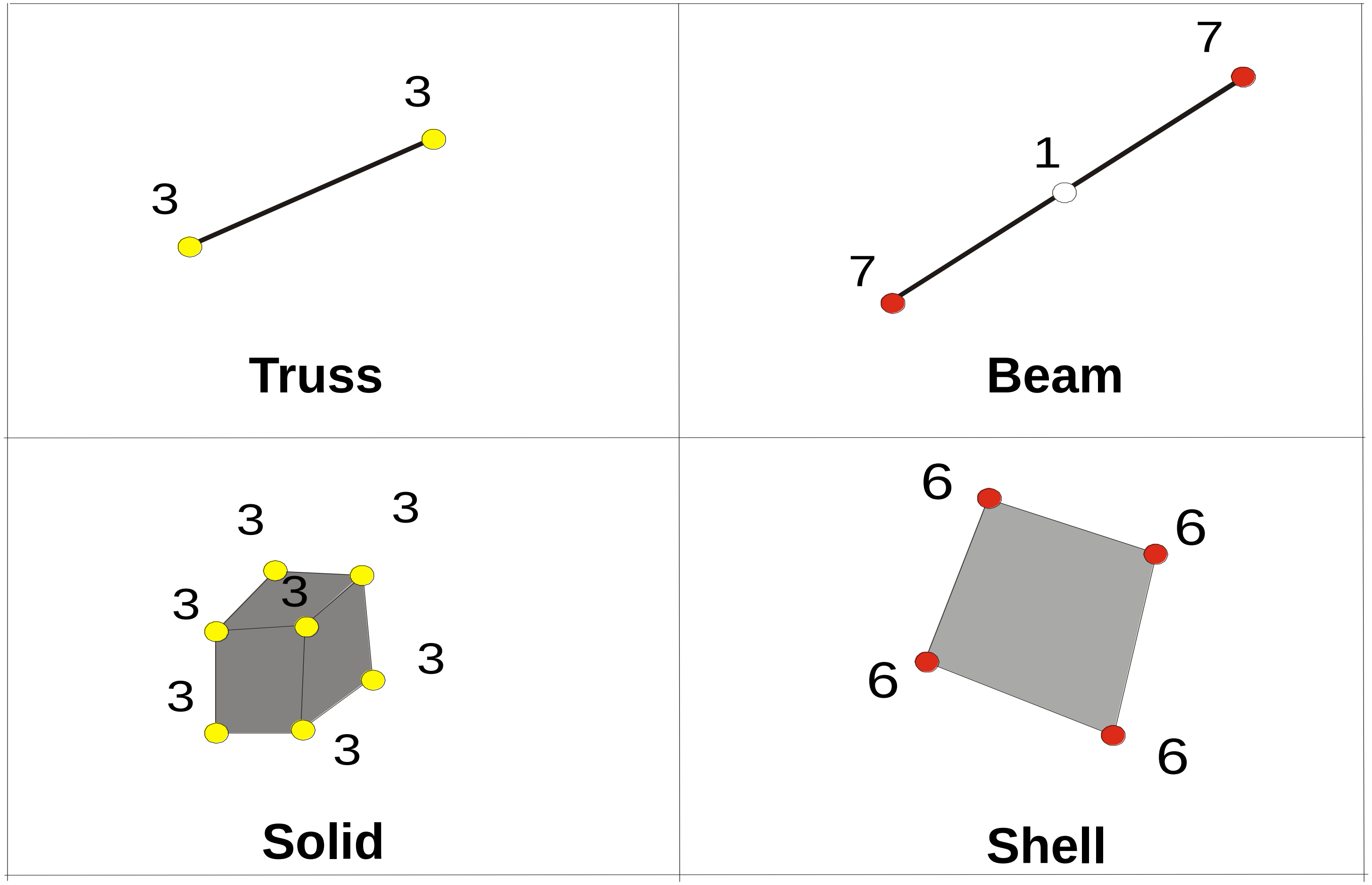
- For **shell elements**, a **uniaxial temperature distribution** is calculated **across the thickness** of the slab using pseudo-2D conductive finite elements. The temperature at the through thickness points of integration for the mechanical analysis is linearly interpolated between the nodal temperatures.





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 and validation
4) User community and references
5) 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. It should be verified when the simulation has been completed that no compressive force is developed in the element during the fire)
  - 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

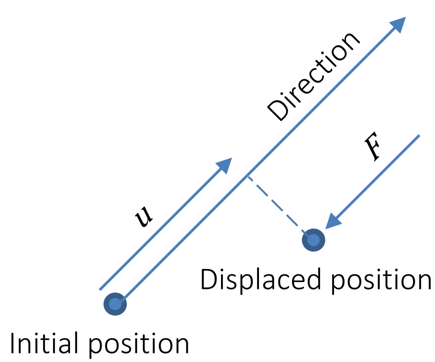
- Quadrangle based on 4 nodes
- 6 DoF at each node: 3 translations and 3 rotations
- Integration on the plane is performed numerically using 4 points of Gauss
- Integration on the thickness is performed numerically with the user choosing the number of Gauss points (from 2 if membrane behavior dominates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

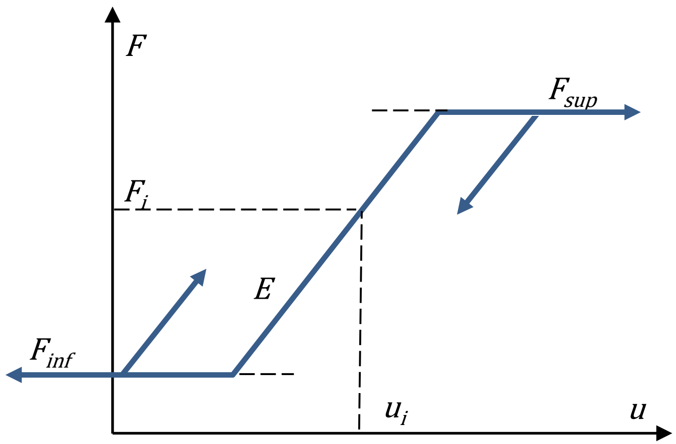
Solid FE

- 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

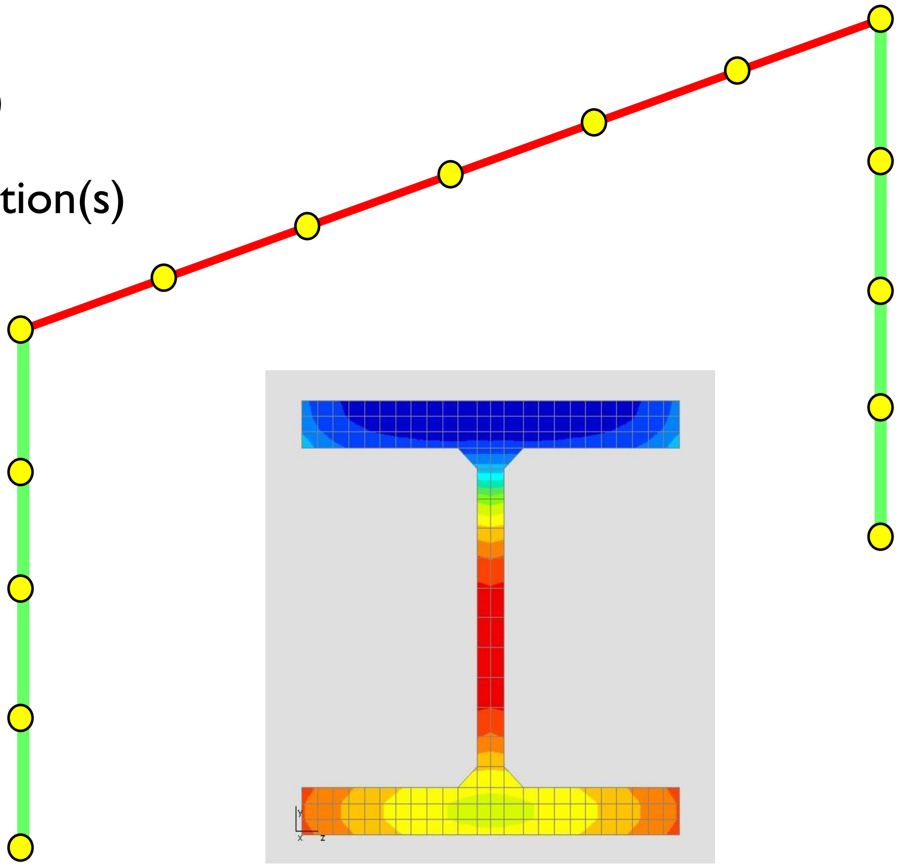




1) Introduction to SAFIR
    a. Description of the software
    b. 3 steps procedure: fire, thermal, mechanical
    c. Selection of Finite Element type for thermal-mechanical analysis
    d. Description of the available FE
    e. **General principle of a thermal-mechanical analysis with beams**
    f.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

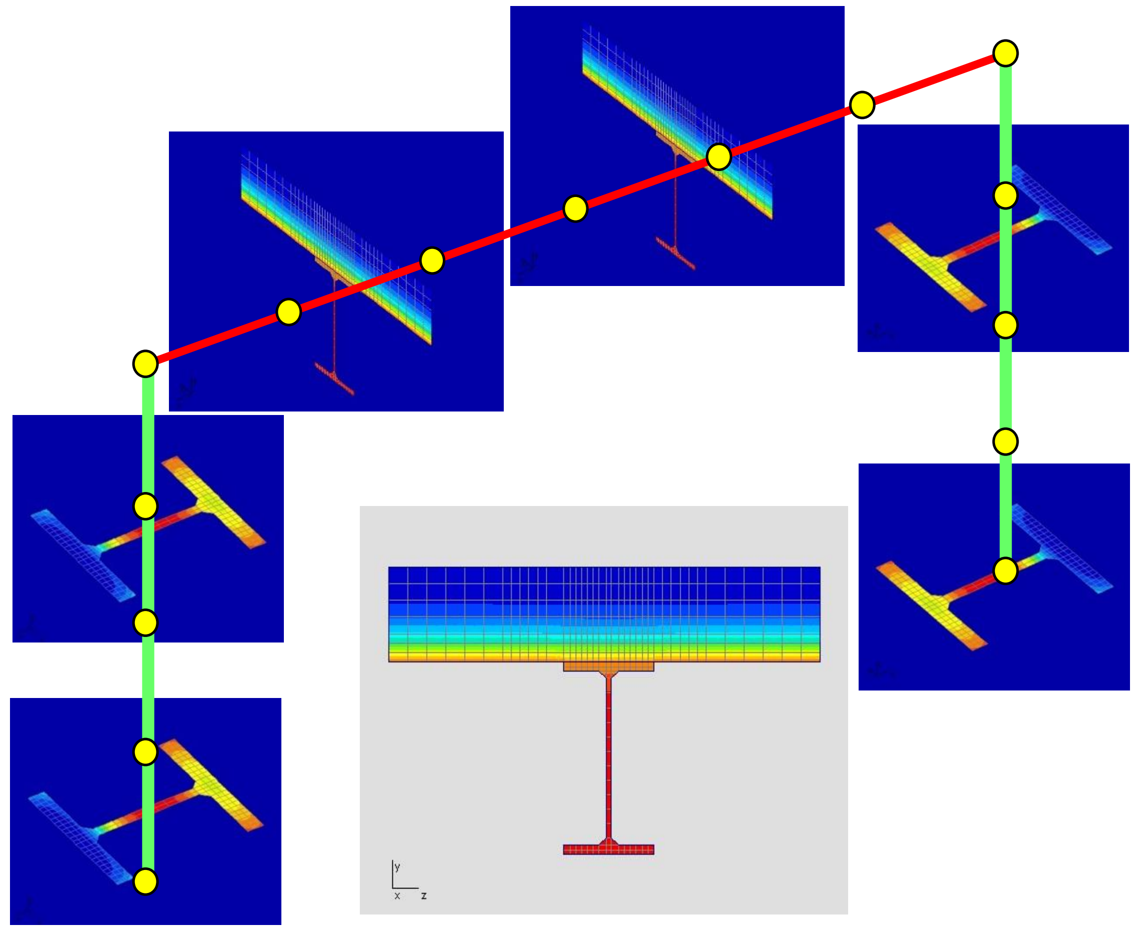
General principle of a mechanical analysis based on beam FE (1/2)

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 and validation
4) User community and references
5) 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		

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 and validation

4) User community and references

5) Resources

Pre-processor GiD

- GiD is a graphic pre-processor
- 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. A free demonstration version can be obtained: http://www.gidhome.com/
- The *problem types* are developed by the SAFIR team to allow generating SAFIR input files from GiD, and can be downloaded for free: https://www.uee.uliege.be/cms/c_2673990/en/safir-free-downloads

Note: use of GiD is by no mean mandatory – 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_2673990/en/safir-free-downloads

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

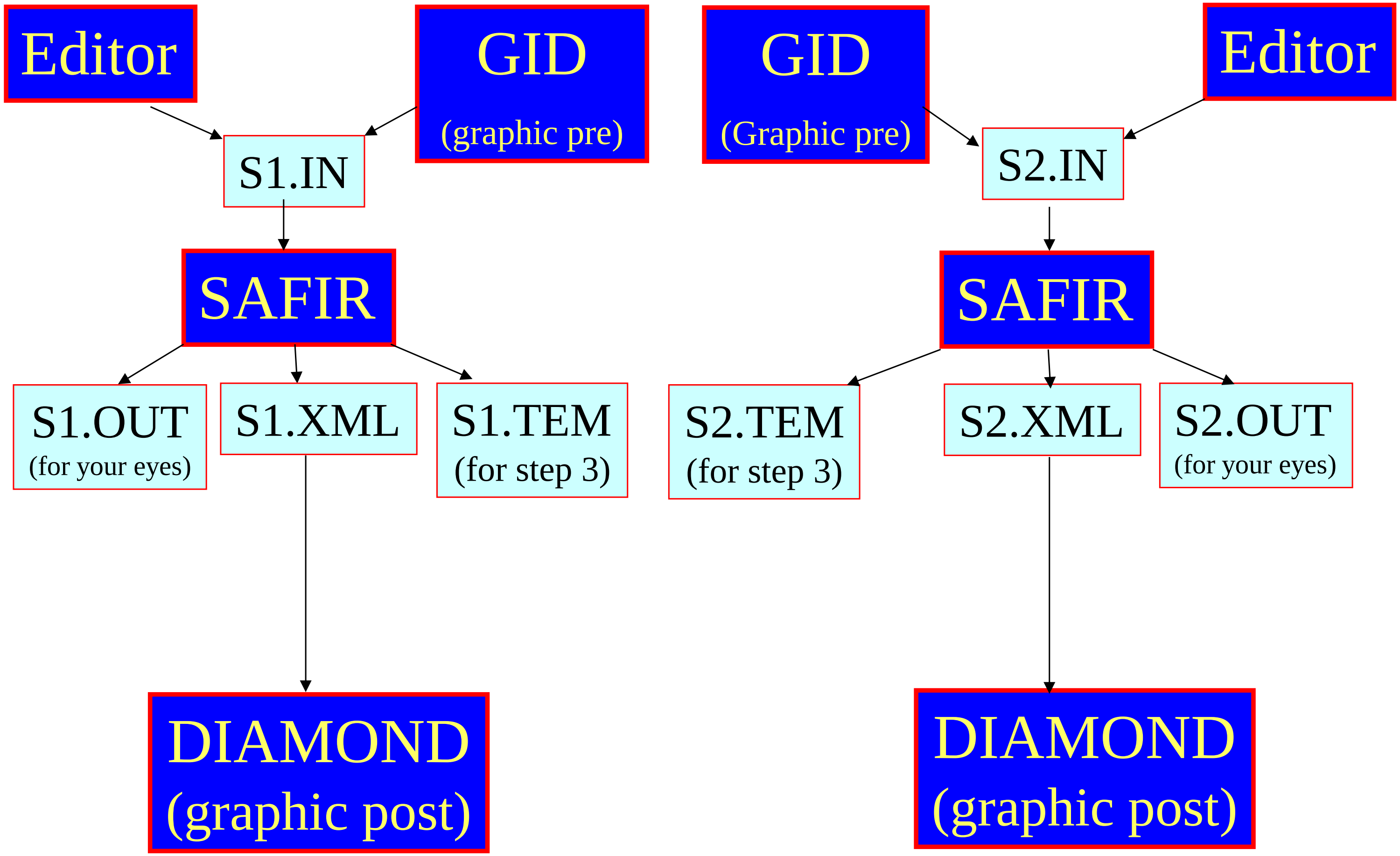
- One mechanical calculation for a structure made of beam elements
- In which there are 2 different section types

Note: one new section type must be considered if:

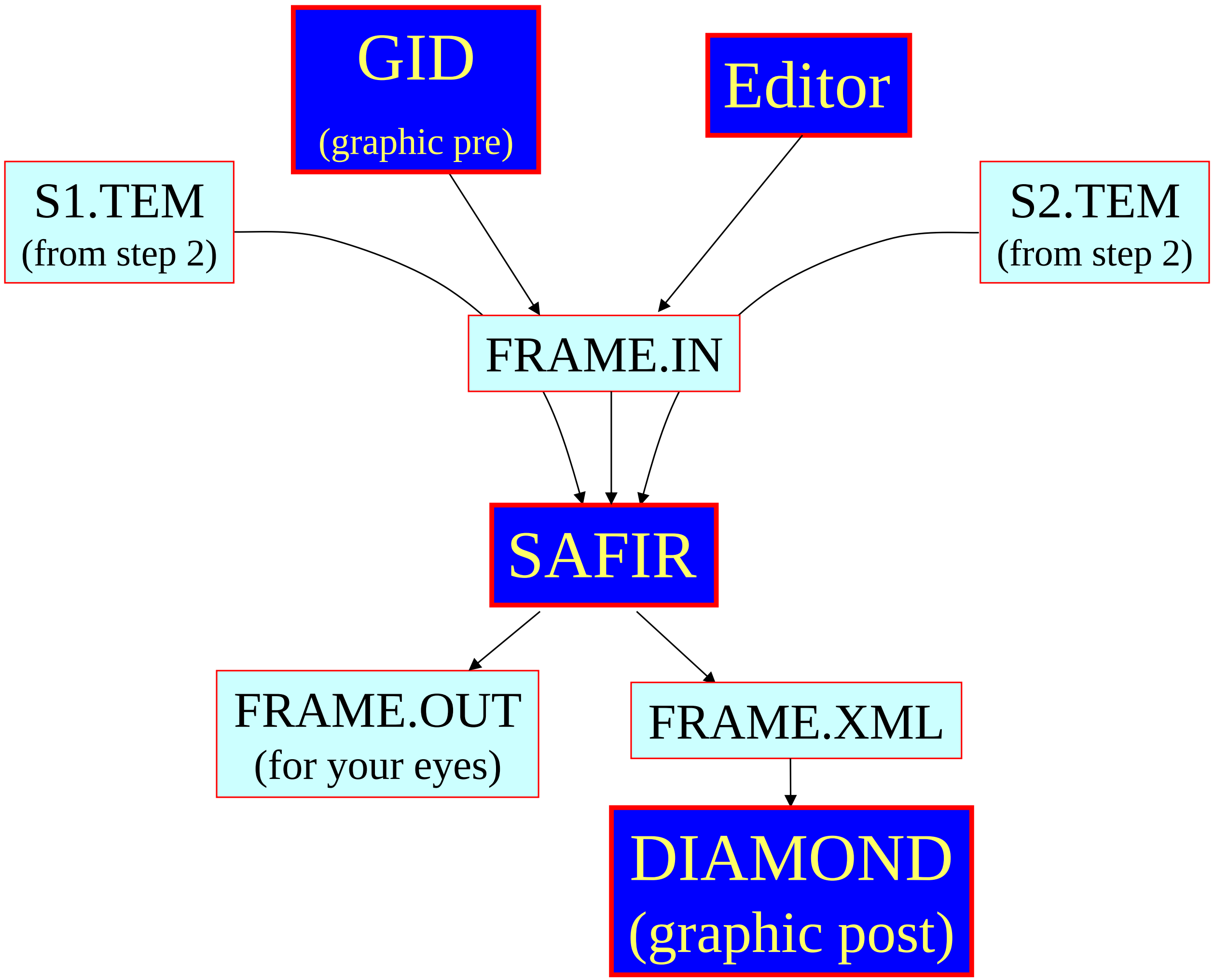
- the geometry of the section is different,
- the fire curve is different,
- the thermal properties are different,
- the mechanical properties are different.

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

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



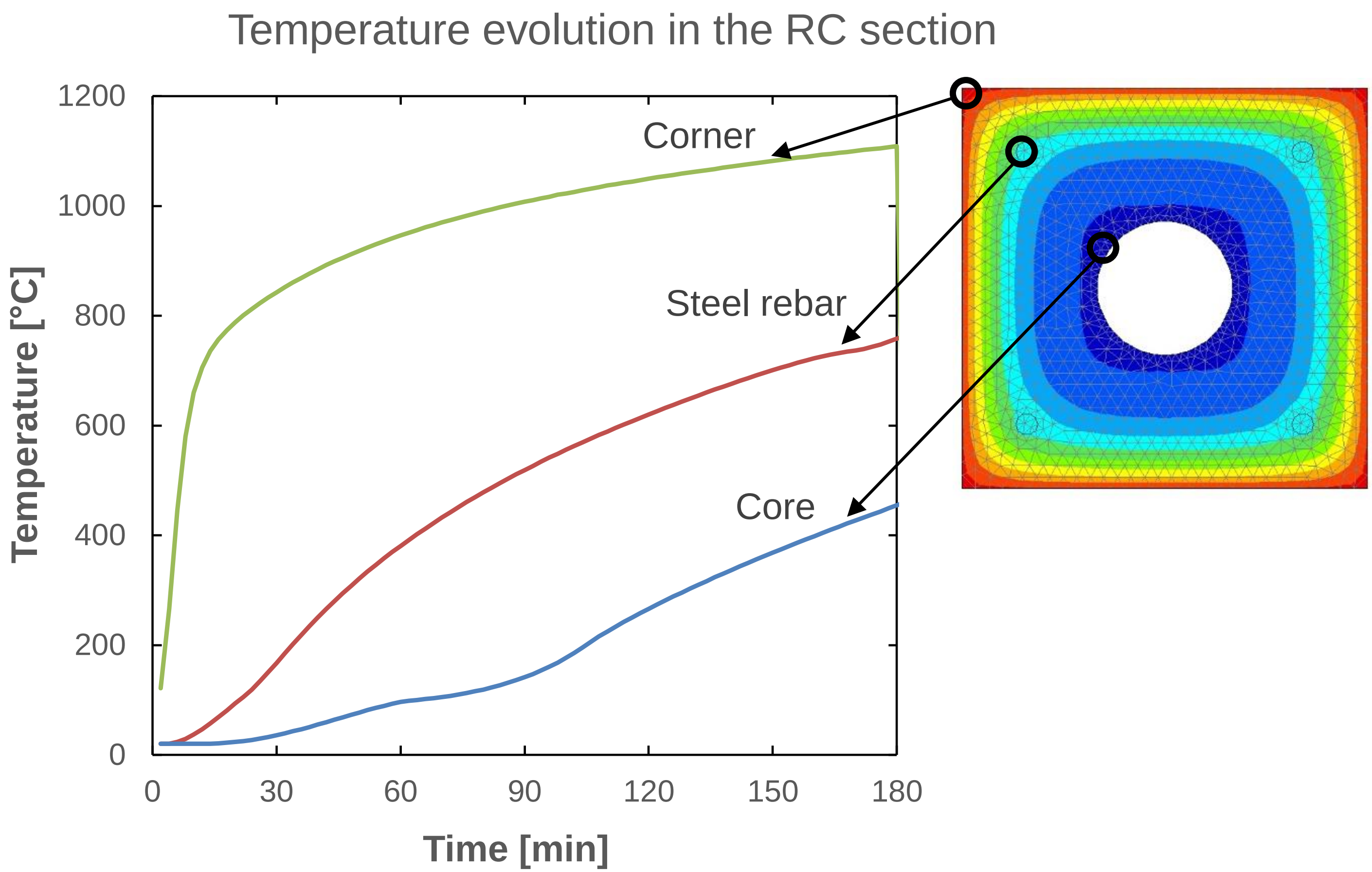
Organization of the files for a typical calculation – Example, Mech. Analysis



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 and validation**
4) User community and references
5) Resources

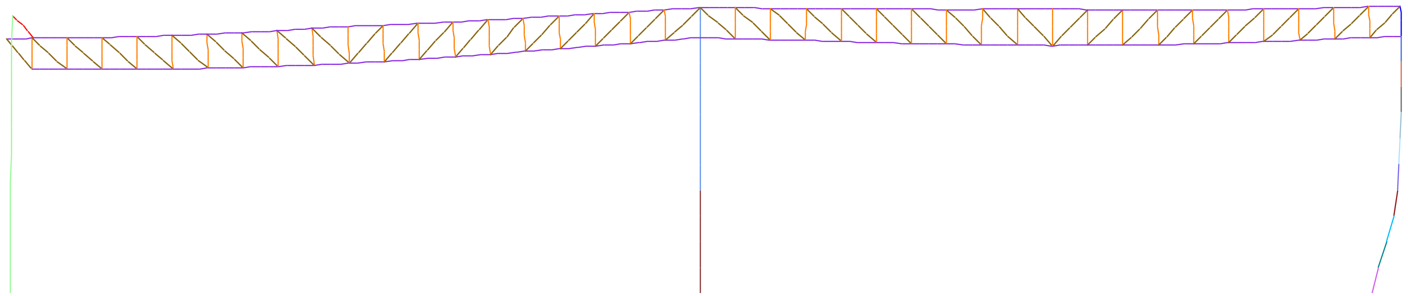
Example of thermal analysis

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



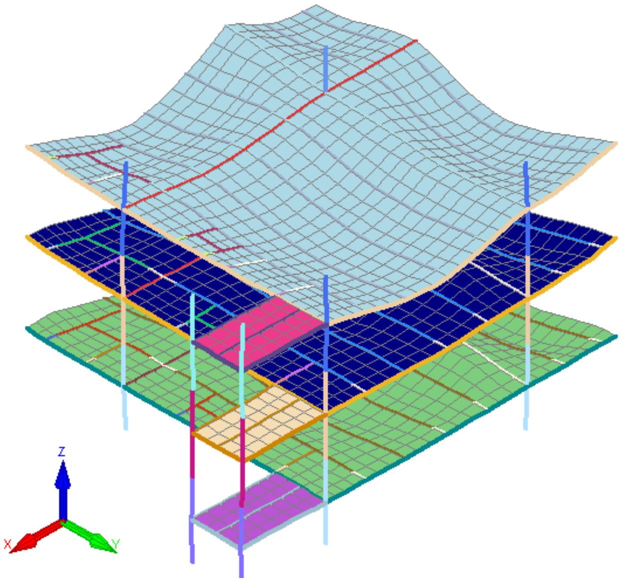
Example of mechanical analysis

2D mechanical calculation – Steel frame



3D mechanical calculation – Steel-concrete building

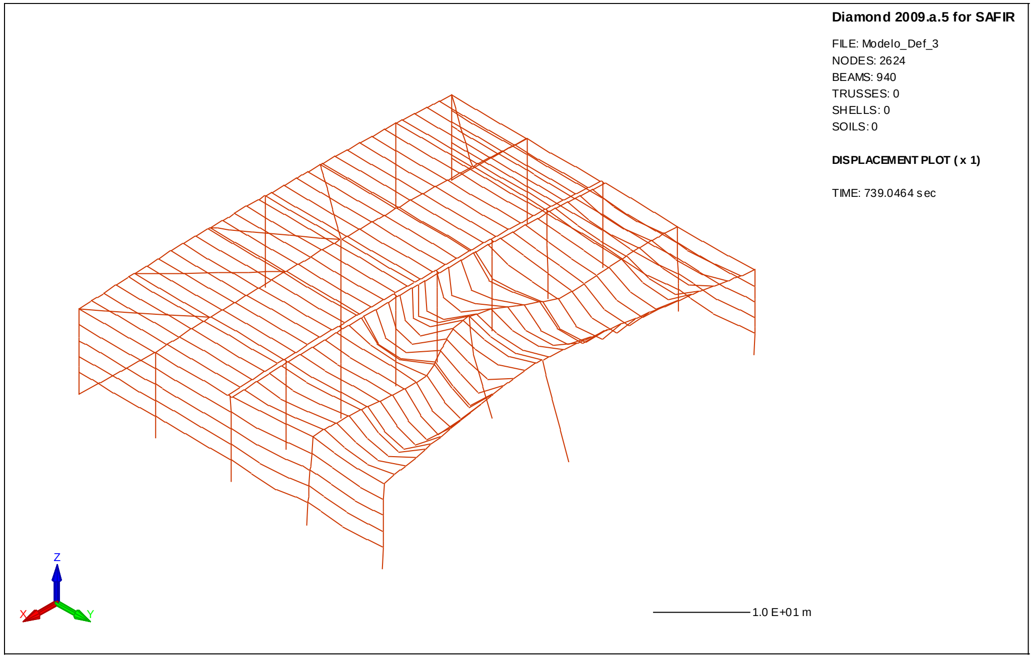
Case study by R. Fike and V. Kodur
Michigan State University, USA
Partial model of an eight story steel frame office building



Example of mechanical analysis

3D mechanical calculation – Steel frame

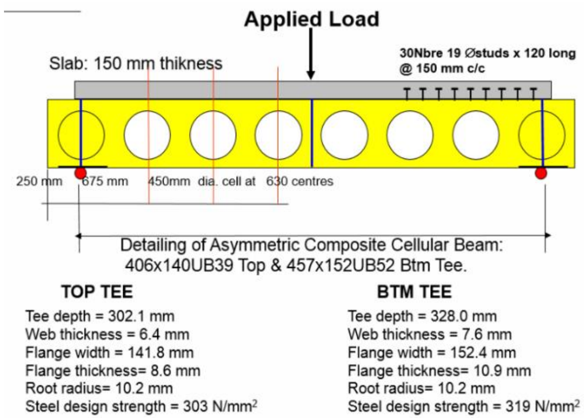
Flumilog test, INERIS, France
2 624 nodes, 940 beam elements

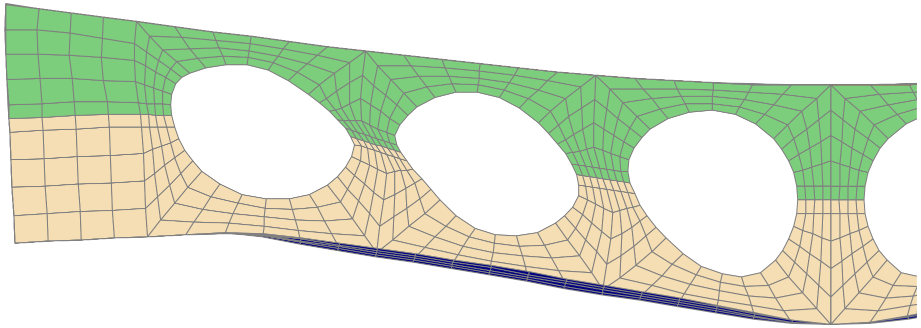


Example of mechanical analysis

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

Experimental test and finite element model

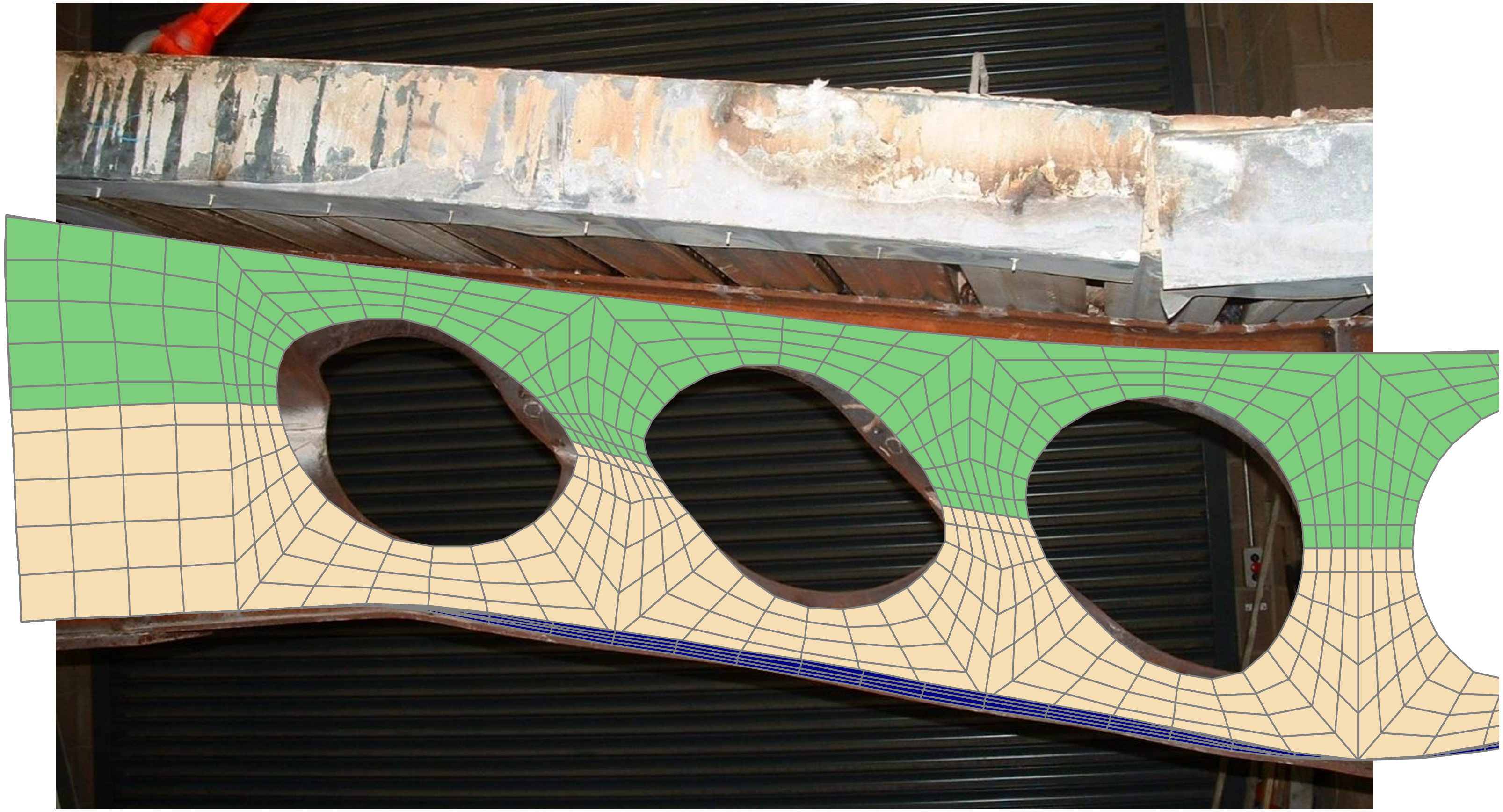




Example of mechanical analysis

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

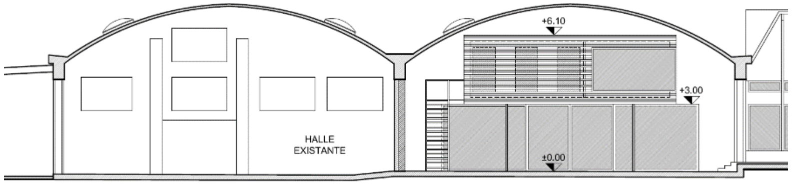
Experimental test and finite element model

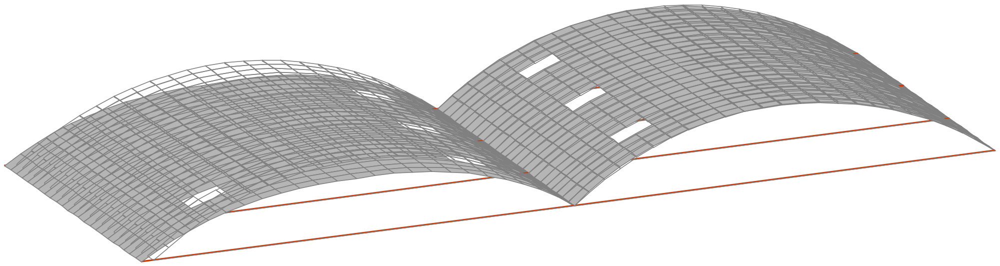


Example of mechanical analysis

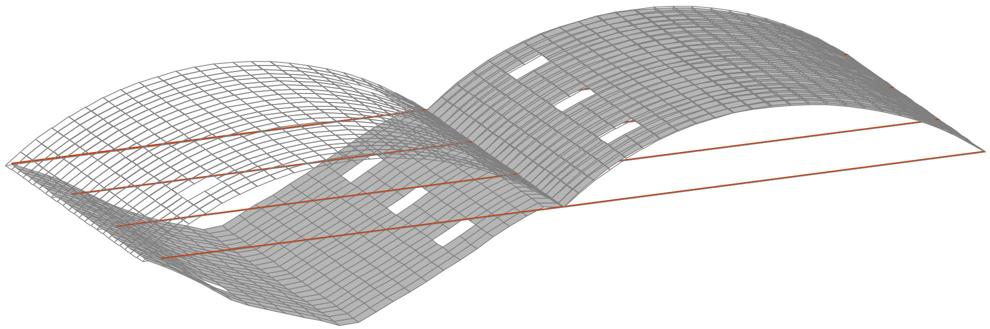
3D mechanical calculation – Structural fire analysis of an arched reinforced concrete roof

Study for ICB, Belgium





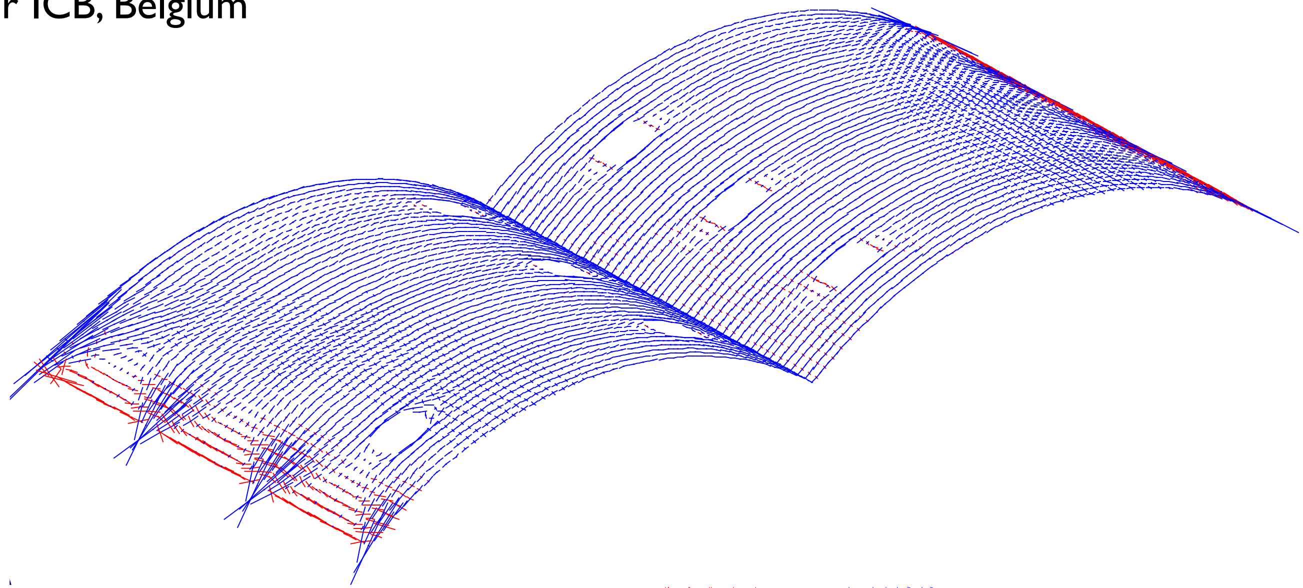
Snap-through collapse under fire when the steel tie rods yield



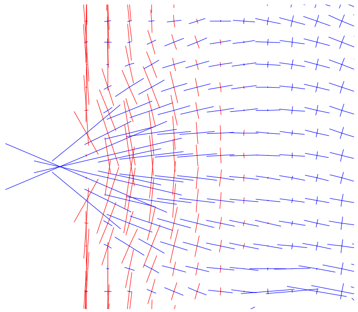
Example of mechanical analysis

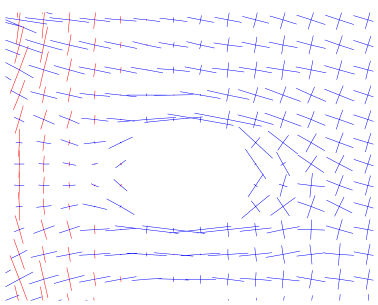
3D mechanical calculation – Structural fire analysis of an arched reinforced concrete roof

Study for ICB, Belgium



Plot of the membrane forces in the concrete shells



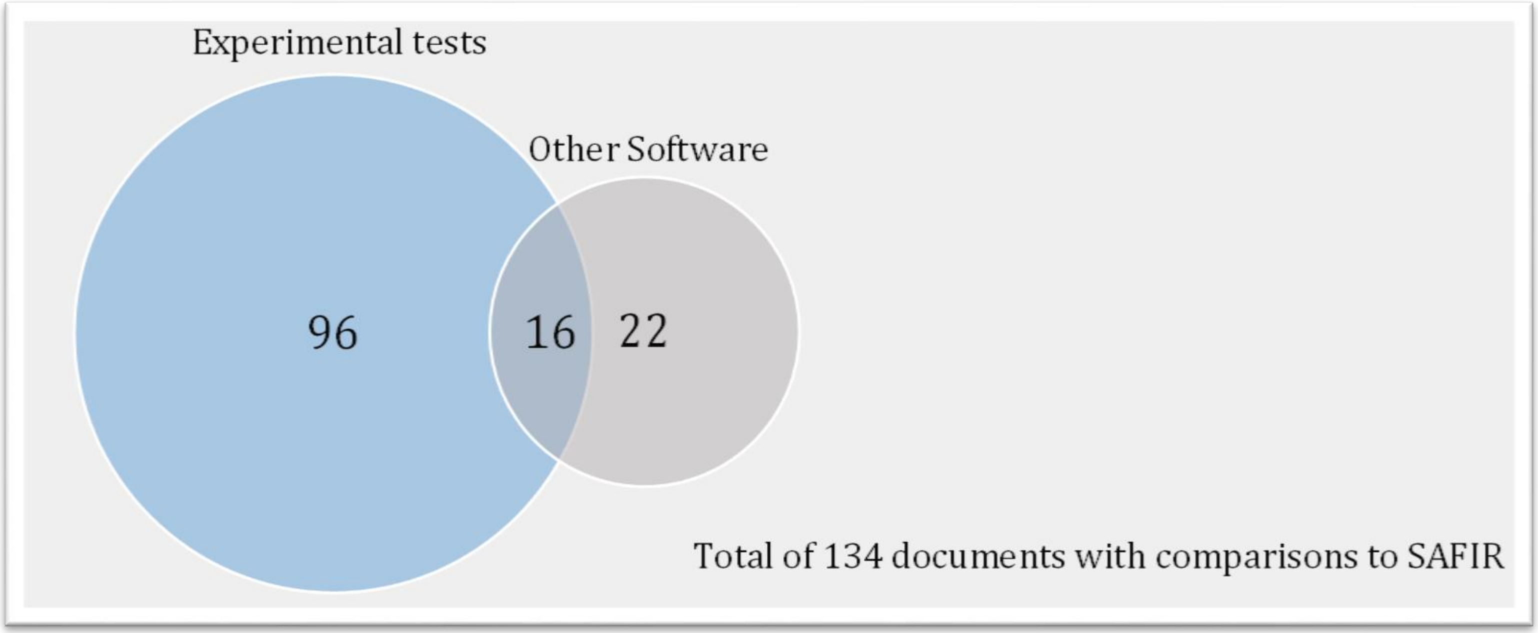


Validation

- 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 http://www.uee.uliege.be/cms/c_2673990/fr/safir-free-downloads

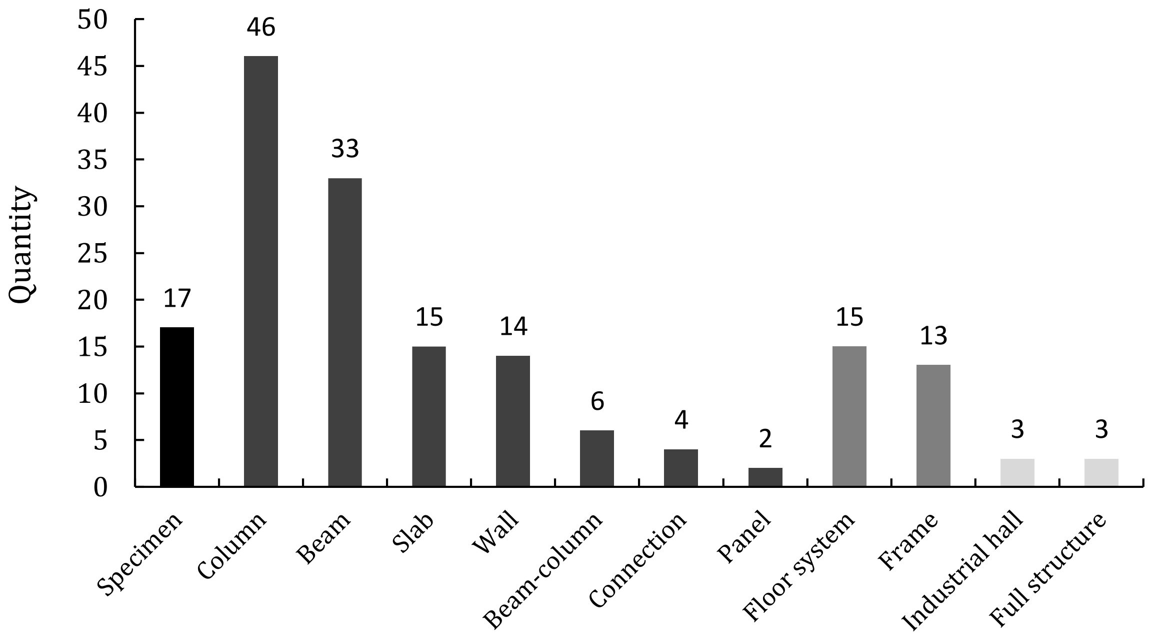
Validation

- 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, including 134 with comparisons between SAFIR and either experimental tests or other software.



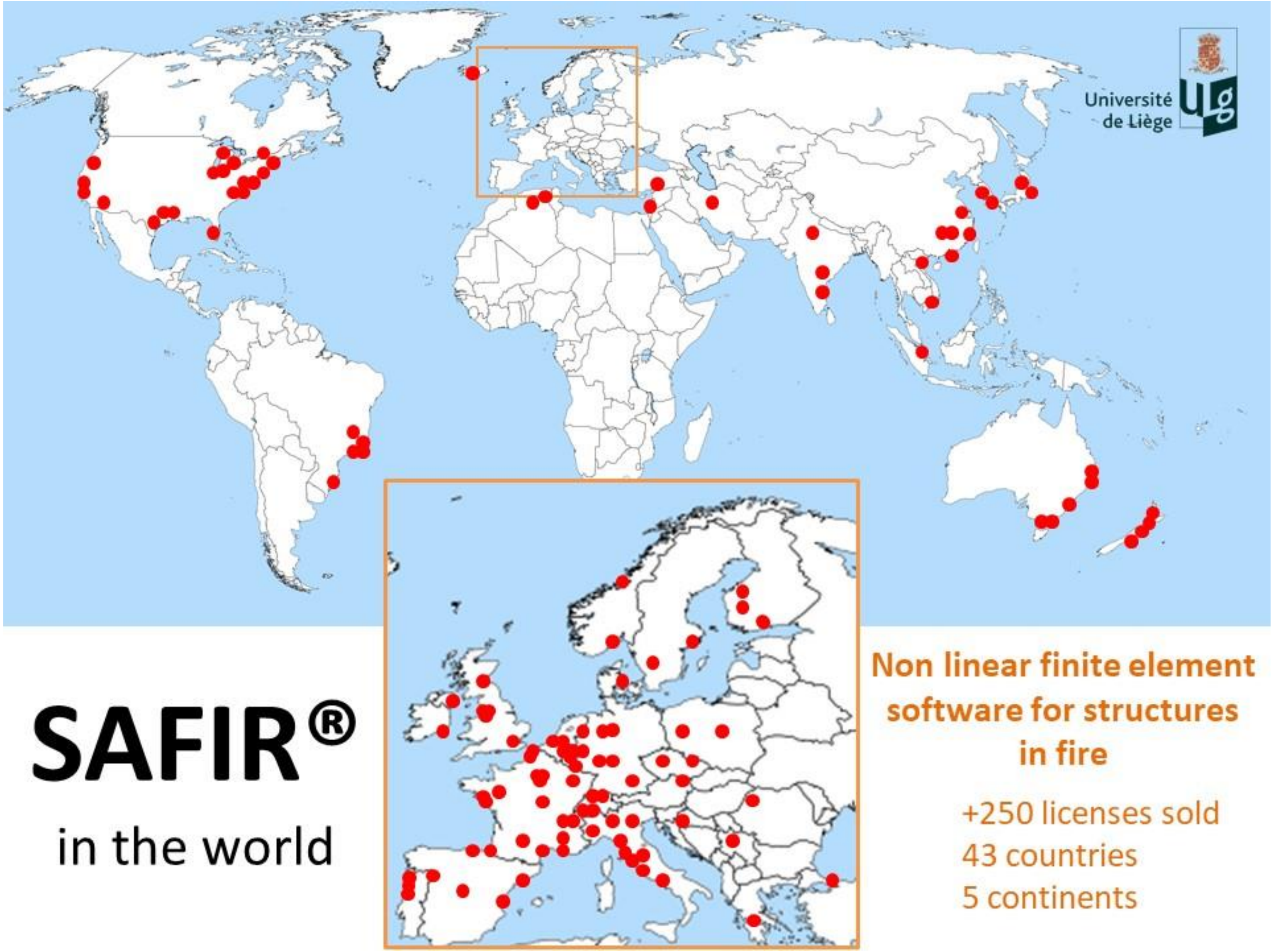
Validation

- 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. We classified them by typology, material, etc.



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 and validation
4) **User community and references**
5) Resources

Users



SAFIR®

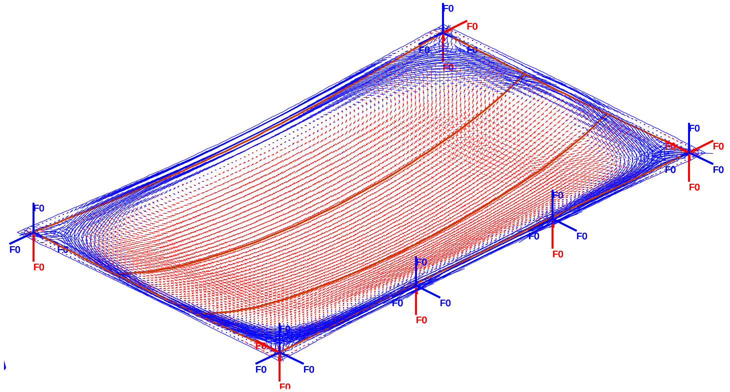
in the world

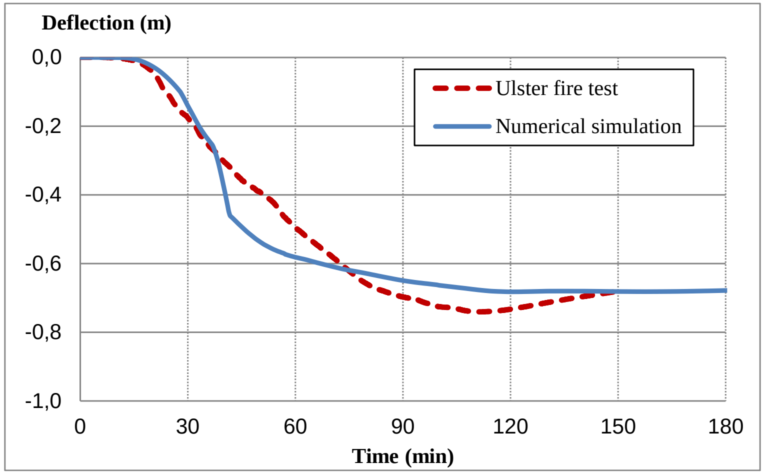
Non linear finite element software for structures in fire

+250 licenses sold
43 countries
5 continents

Example of scientific reference

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

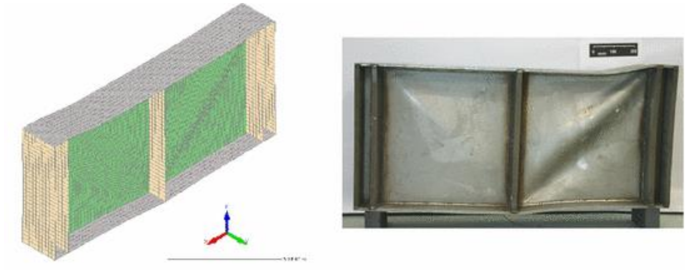




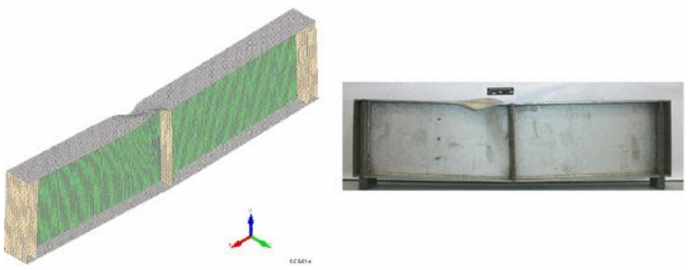
Vassart et al. (2012). *Structures and Buildings, 165*(7), 327–334

Example of scientific reference

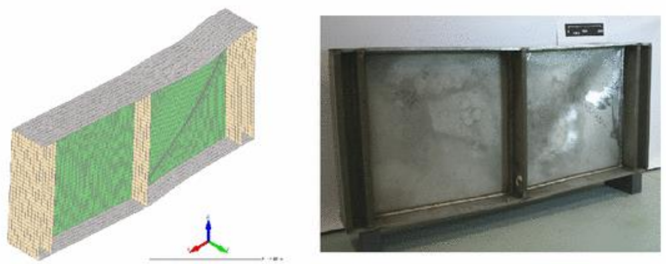
Reis et al. – *Shear buckling of plate girders at normal and high temperature*



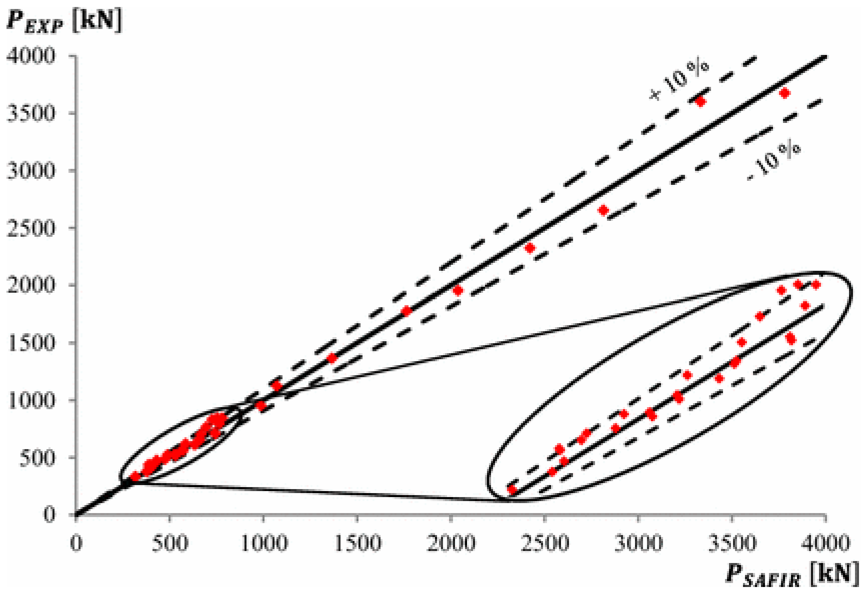
(a) PG20



(b) PG27



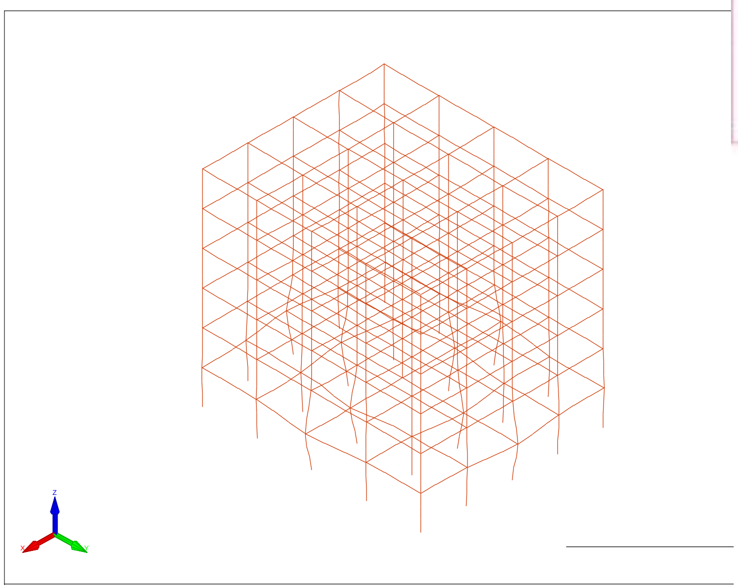
(c) PG22

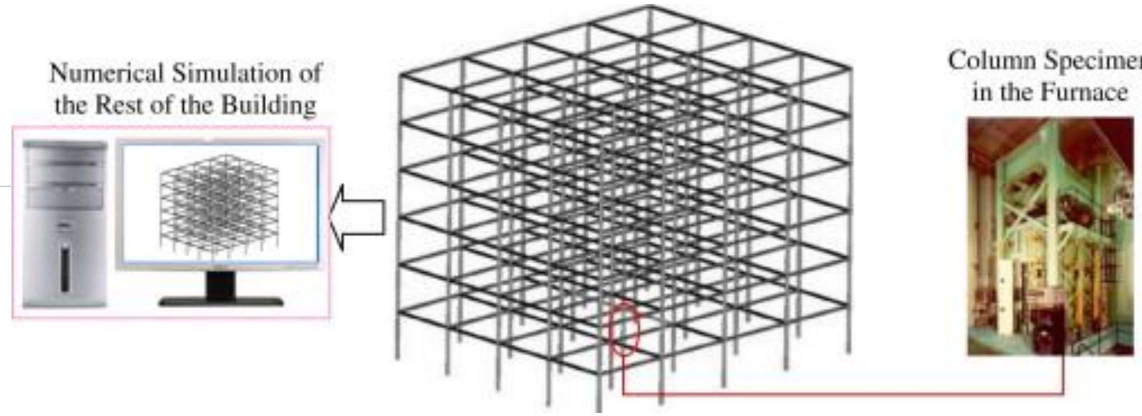


Reis A., et al. (2017) *Fire technology* 53.2: 815-843.

Example of scientific reference

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





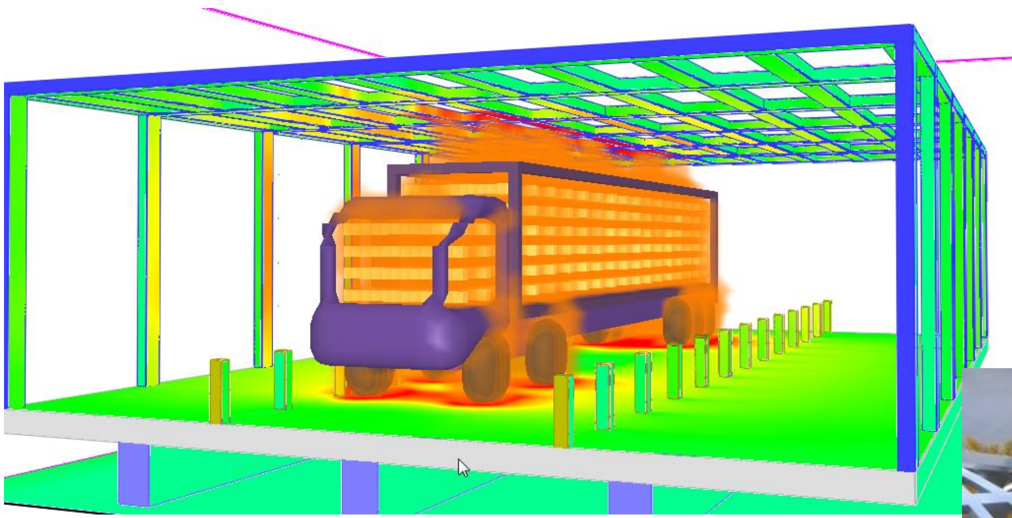
Use of SAFIR for HFT:

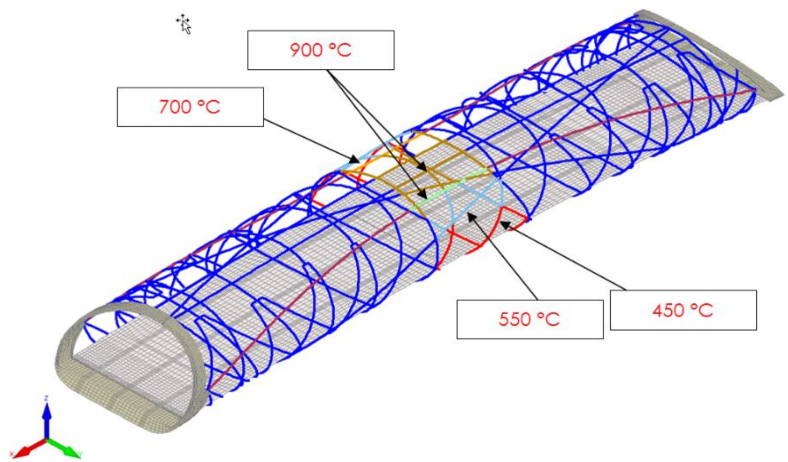
Mostafaei, H. (2013). *Fire safety journal*, *56*, 30-38.

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

Example of commercial reference

MP Ingénieurs Conseils, Switzerland – Wilsdorf bridge in Geneva

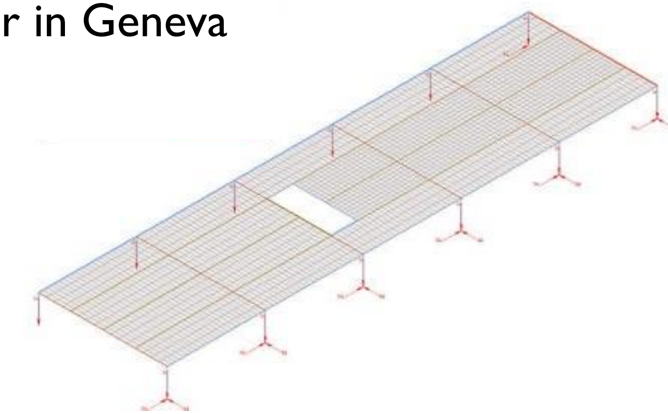




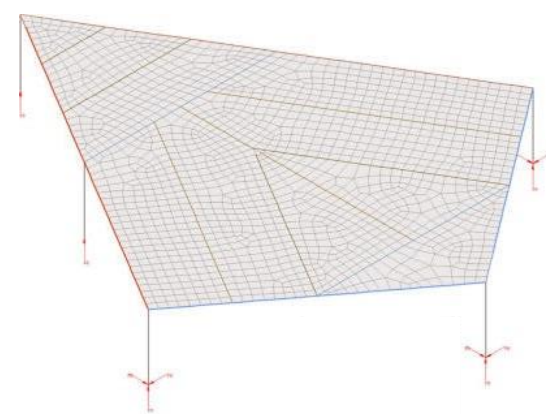
Tonicello, E. et al. (2012). *Proc. 7th Structures in Fire Conf.*

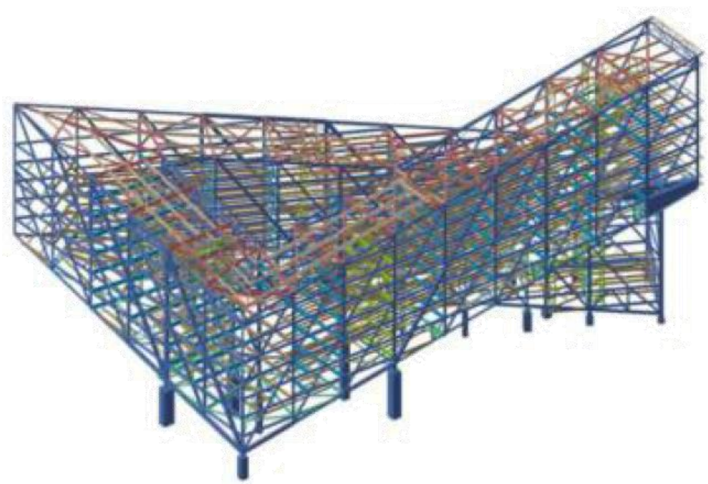
Example of commercial reference

INGENI, Switzerland – Japan Tobacco Headquarter in Geneva



SAFIR models for fire analysis





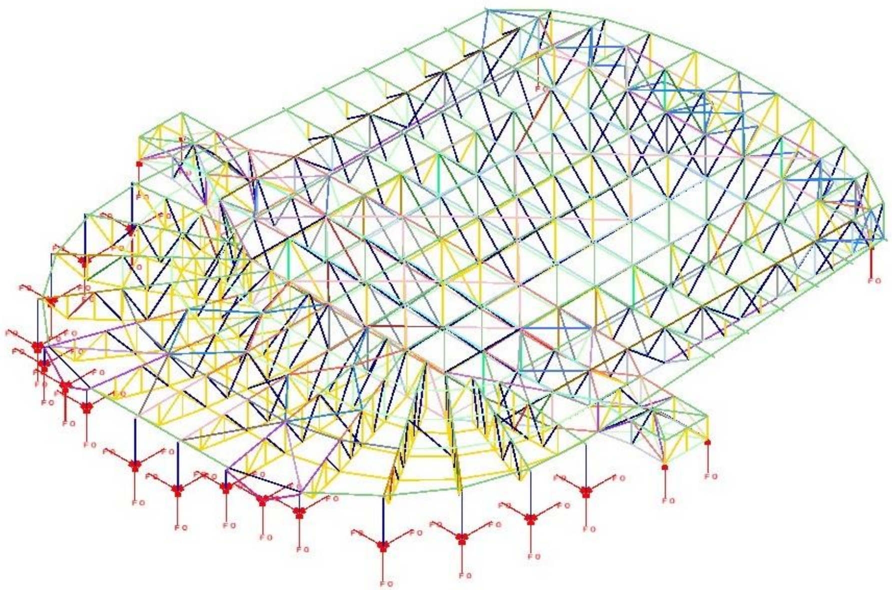
Model for ambient temperature analysis (not SAFIR)

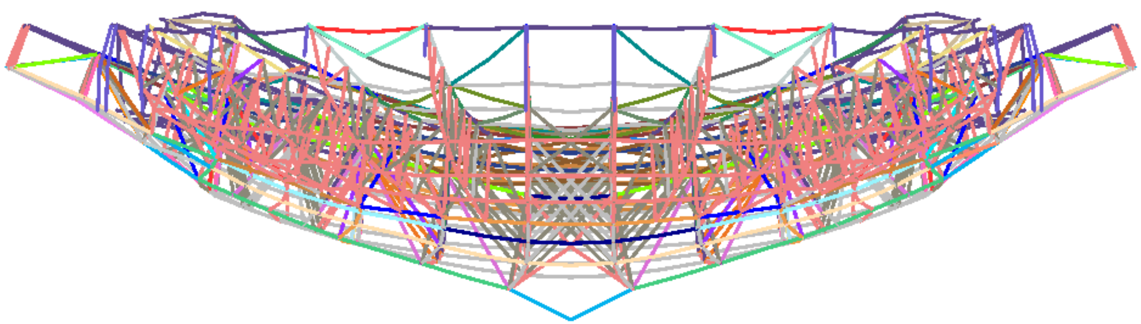
Lelli, L., Loutan, J. (2017). *Journal of Structural Fire Engineering*.

Example of commercial reference

BuroHappold Engineering, UK

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





Scientific references

Please quote the following paper in any work which made use of 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. http://hdl.handle.net/2268/202859

Additional relevant references include a 2005 AISC publication (+470 citations on Google Scholar as of May 2019) as well as many research papers, some of which are listed here: https://www.uee.uliege.be/cms/c_2673993/en/safir-publications

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 and validation
4) User community and references
5) **Resources**

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
- More examples: https://mars.jhu.edu/safir/worked-examples/
- Contact us at safir@uliege.be
  - When emailing, please make sure to mention your organization, type of license, and describe the problem in details (possibly attaching the input files)

```
----- Mail original -----
De: [redacted]
À: safir@uliege.be
Envoyé: Mardi 28 Janvier 2020 09:17:43
Objet: Safir problem

Dear Safir staff,
I've been trying for days to solve this column.
Could you please help me?

Best regards,
S[redacted] E[redacted]
```

Note: A GID folder was attached to this e-mail

Conditions

Licenses: visit https://www.gesval.be/catalogue/safir-2019-commercial

✓Academic license: 1200 € + taxes.

✓Commercial license: 6000 € + 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)

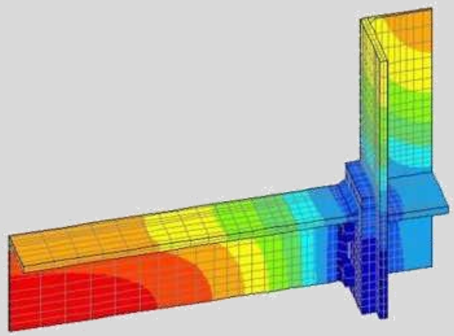
✓Free updates during 1 year

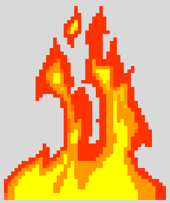
Training sessions

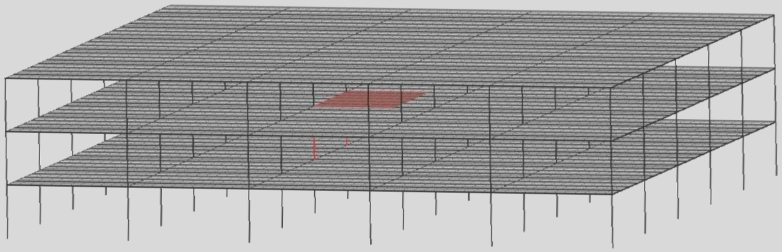
✓Organized regularly at Liege University and JHU

✓Organized on demand

- Can be organized on site at the client's







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.