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

General features of mechanical analyses

1) Time integration procedure

2) Comeback strategy and time steps

3) Loads

4) How is failure considered in the software?

General features of mechanical analyses

1) Time integration procedure

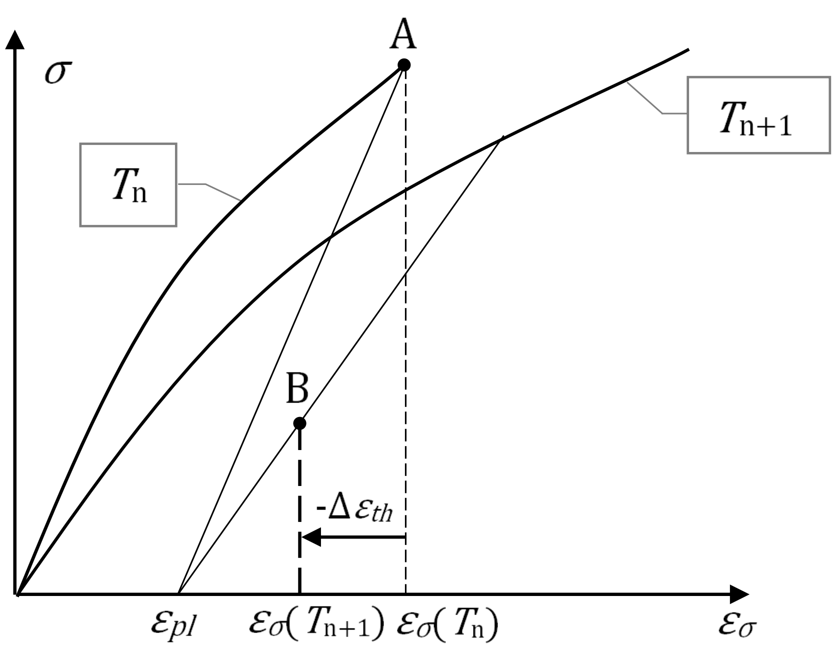
2) Comeback strategy and time steps

3) Loads

4) How is failure considered in the software?

Time integration procedure in mechanical analysis

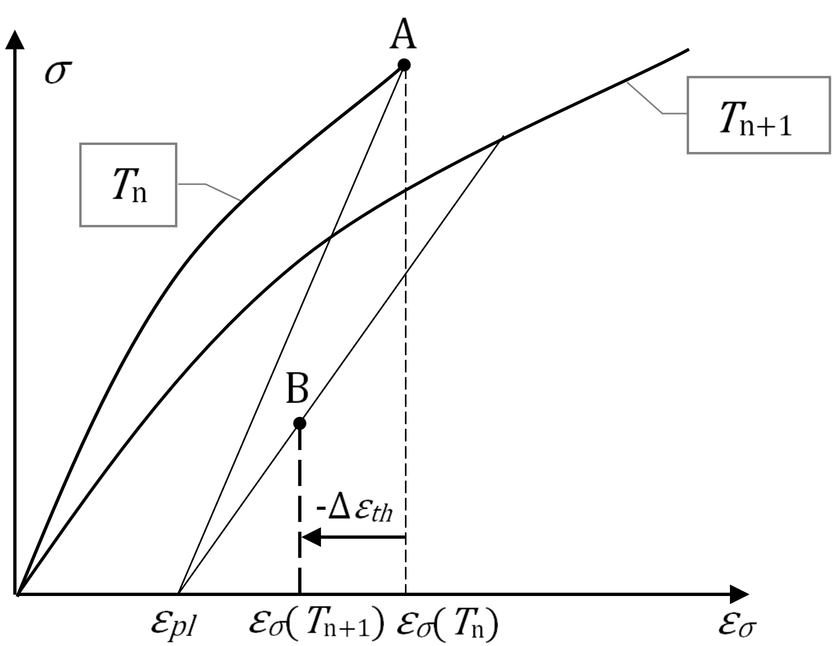
- Iterative procedure to integrate in time from one converged time step n (point A) to the next step n+1.
- At converged time step n, the stress-related strain is $\varepsilon_\sigma(T_n)$ and the plastic strain ε_{pl}.



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.

Time integration procedure in mechanical analysis

- 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}.

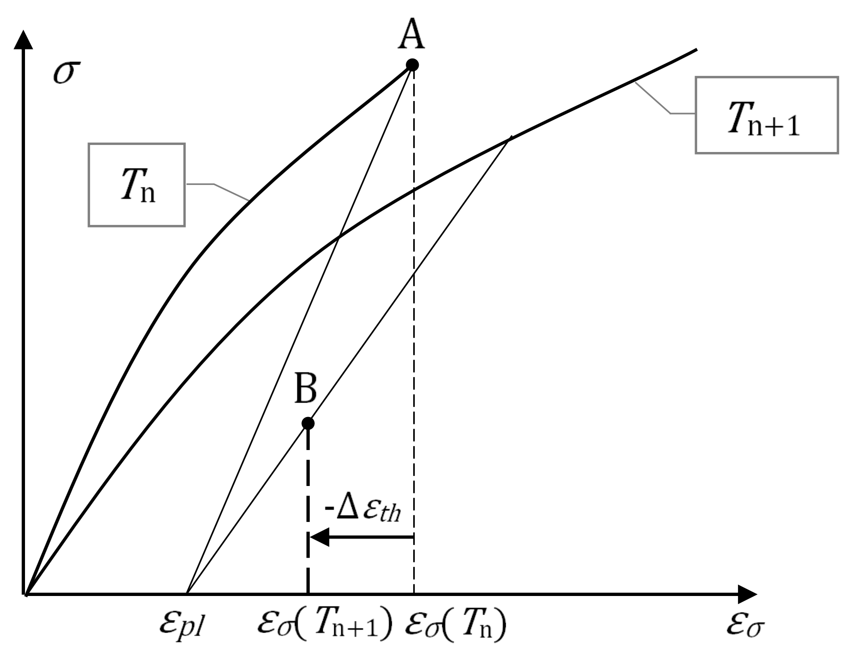


4. With displacements "frozen", ε_{tot} are constant. So the $\Delta\varepsilon_{th}$ generates a new value of stress-related strain, $\varepsilon_\sigma(T_{n+1})$. For a temperature increase, the structure is "artificially compressed".

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).

Time integration procedure in mechanical analysis

- 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}.

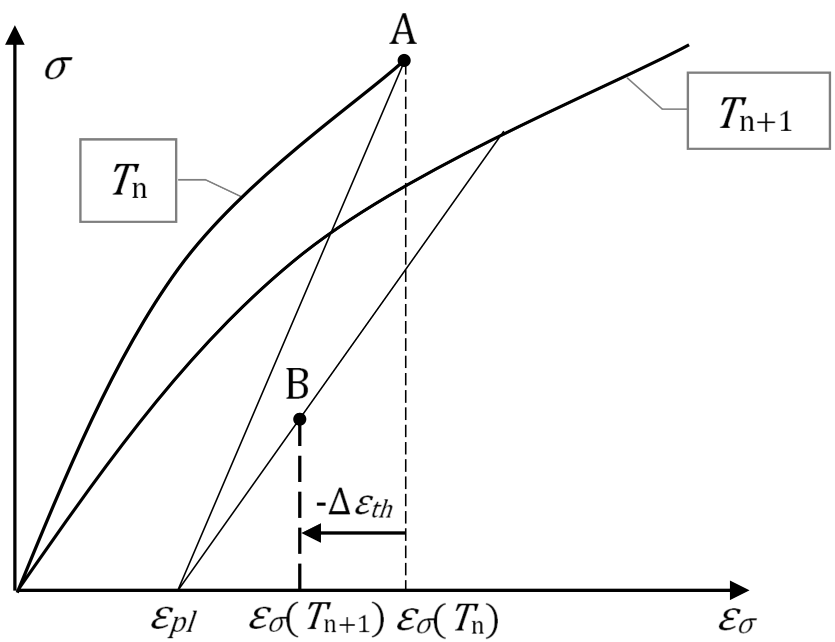


6. The stresses are integrated on the volume of the elements to compute the internal nodal forces which are not anymore in equilibrium with the applied nodal forces. In case of temperature increase, the structure undergoes an internal state of out of balance compression.

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

Time integration procedure in mechanical analysis

- 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.

1) Time integration procedure

2) Comeback strategy and time steps

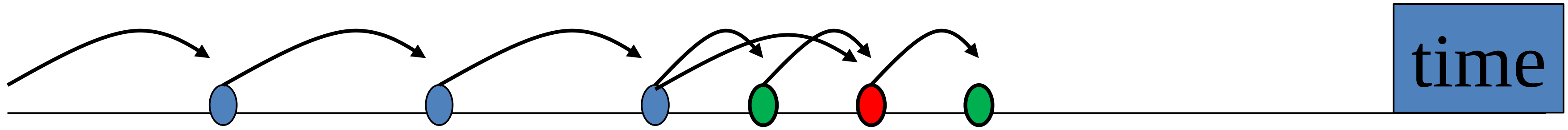
3) Loads

4) How is failure considered in the software?

Mechanical analyses are nowadays performed in dynamic mode, in order to reduce problems of local and temporary failure.

```
COMEBACK      0.0001
```

seconds



COMEBACK procedure stops when the time step becomes smaller than [0.0001] seconds

```
TIME
0.1024     3600.     10.
```

Max. value of the time step

End of the simulation

Value of the first time step

1) Time integration procedure

2) Comeback strategy and time steps

3) Loads

4) How is failure considered in the software?

NLOAD n

1 load vector =

several nodal loads + several distributed loads

multiplied by a function of time
(F1 = 1, F1PS = t, FLOAD = t/20 ≤ 1, etc)

1) Time integration procedure

2) Comeback strategy and time steps

3) Loads

4) How is failure considered in the software?

BIG QUESTION

How is failure considered in the general calculation model?
(in SAFIR / structural fire engineering software / …) ?

ANSWER

- 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?

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

Build a model and run SAFIR

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)
- Load level (load is too big)
- Time step (too big)
- …

Check the **last lines of the .OUT** text file for an indication!

Case II: the software runs (and it will run as far as possible)

⇒ Examine the results

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

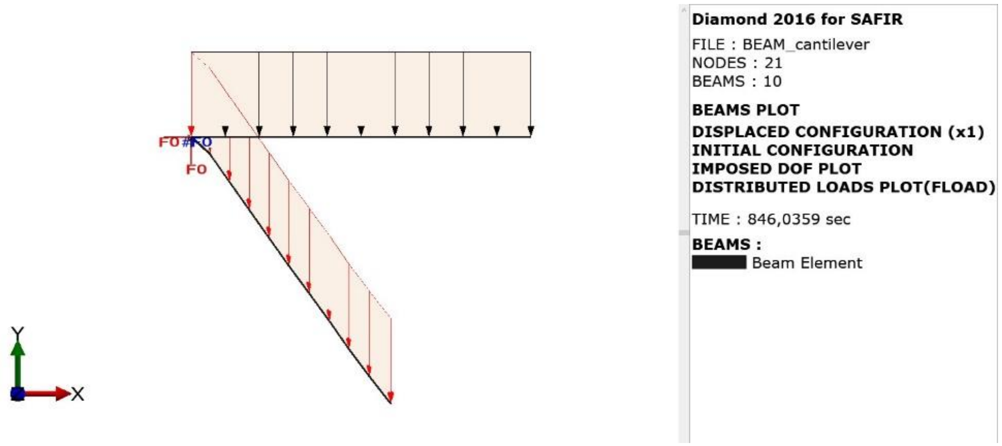
<u>Case II</u>: the software runs

⇒ 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



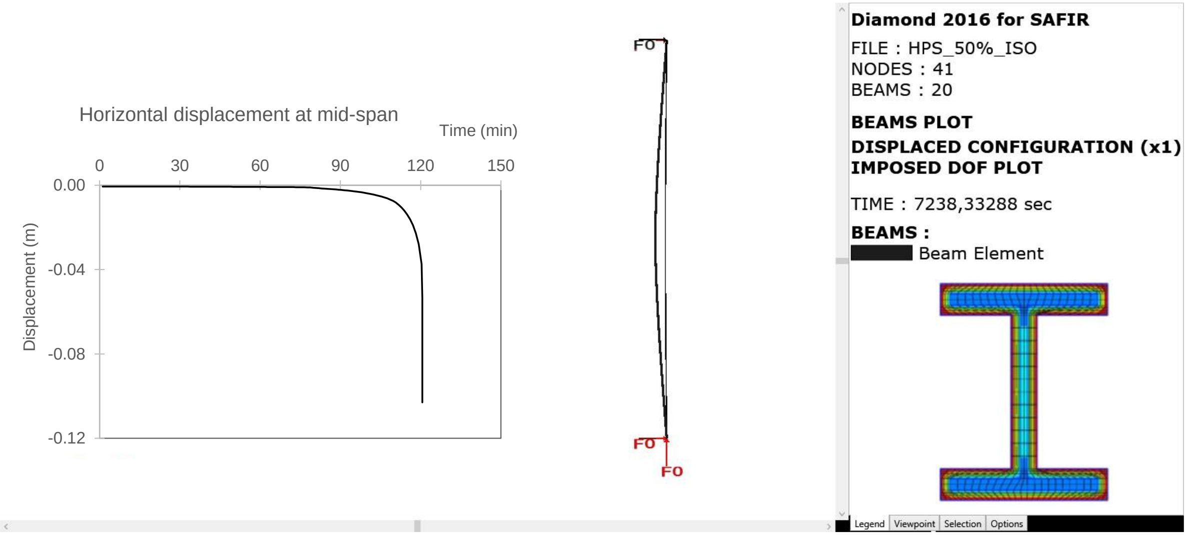
So in practice, how to assess failure from an advanced (numerical) analysis?

<u>Case II</u>: the software runs

⇒ Examine the results

B. The displacements are not too big

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



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

Case II: the software runs

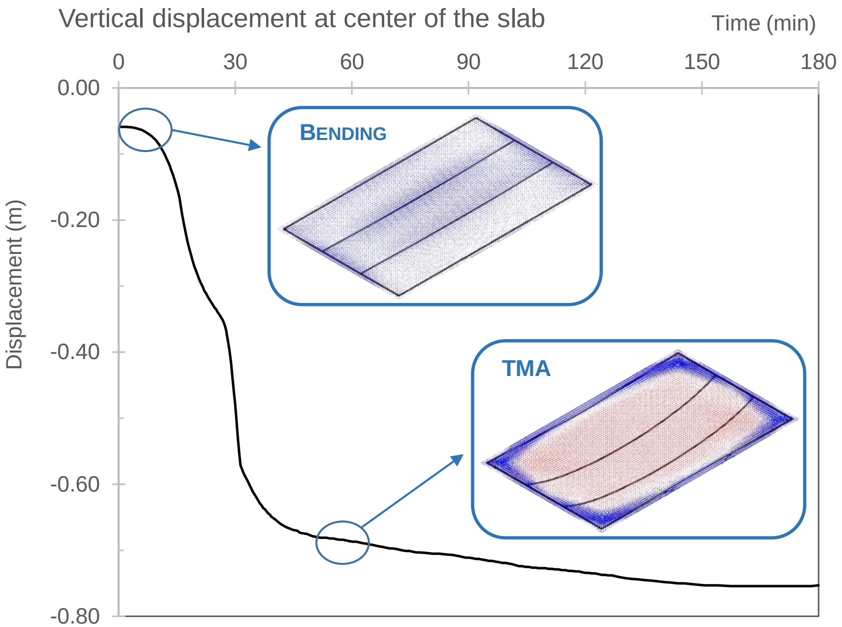
⇒ Examine the results

B. The displacements are not too big

⇒ Look for vertical asymptote in the displacement curve of (at least) 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; …



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

<u>Case II</u>: the software runs

$\Rightarrow$ Examine the results

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

$\Rightarrow$ This is a good indication of numerical failure (premature lack of convergence)

<u>Note</u>: except when the failure mode is really fragile (e.g. timber)

- 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.

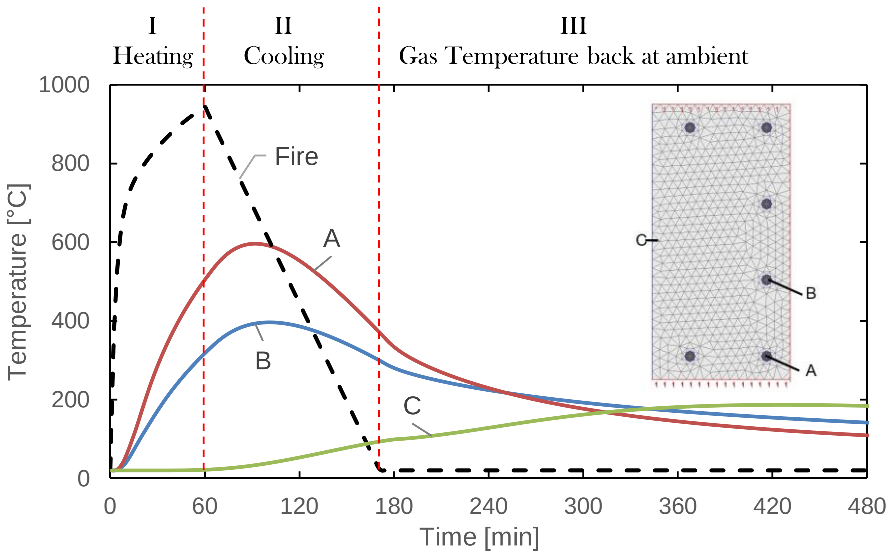
Again, check the **last lines of the .OUT** text file for an indication! For instance, "steel in descending branch" is a proxy for steel fracture (strain exceeded the ultimate value).

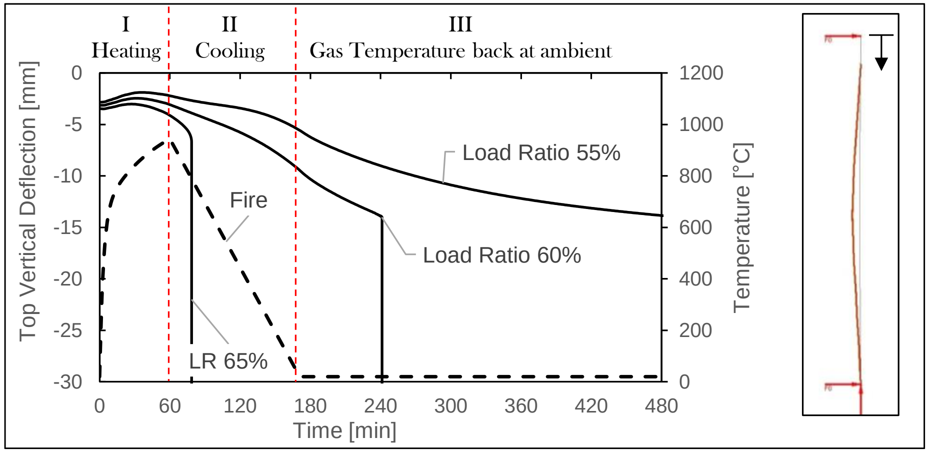
So in practice, how to assess failure from an advanced (numerical) analysis?

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.