# A new structural behavior to perform efficient nonlinear SFR fuel bundle thermomechanical analysis

Bertrand Leturcq, Jean-Baptiste Minne, François Di Paola

# A new structural behavior to perform efficient nonlinear SFR fuel bundle thermomechanical analysis

Bertrand LETURCQ

Atomic Energy Commission (CEA)
DEN, DANS, DM2S, SEMT, Mechanics and Systems Simulation Laboratory (LM2S)
F-91191, Gif sur Yvette, France

Jean-Baptiste MINNE

Communication et Systèmes (CS-SI)
Energy, Industry and Services BU
22 avenue Galilée, 92350 Le Plessis Robinson, France

François DI PAOLA

Atomic Energy Commission (CEA)
DEN, DANS, DM2S, SEMT, Mechanics and Systems Simulation Laboratory (LM2S)
F-91191, Gif sur Yvette, France

A sodium fast reactor (SFR) fuel bundle is usually a strong hexagonal tube containing around 200 fuel pins made of steel. These pins should not touch each other to avoid overheating and damage of the bundle, which constitutes the first confinement barrier. For that purpose a steel wire is often wrapped around each pin from the bottom to the top with a helical shape. This technology maintains a distance between pins and ensures a proper mixing of the sodium, two necessary conditions so as to avoid hot points and damage. Nevertheless the intense fast neutron irradiation induces a significant isotropic swelling and creeping of the pins. If continued, this progressively generates contact closures (phase 1), then mainly a helical bow of the pin (phase 2), and in the end, a strong ovalisation of the cladding sections (phase 3). At such an interaction level, there is a possibility of cladding damage due to thermal creep accumulation.

Several 3D finite elements models of the fuel bundle have been developed including the tube, all the 217 pins and their wrapped wires, at least 7.000 contact, with thermal and irradiation creeping laws. If conventional solid or even shell elements are used, then the huge memory cost and CPU time become completely inconsistent with the project expectations. The first analyses have shown that it was necessary to reduce the number of degree of freedom, but also, to keep the representation of all the pins and all the contact areas. Thus, a new approach is used here. First, the helical deformation of each pin is captured by a simple viscoplastic vertical beam; second, a new connecting element is developed and positioned horizontally on every potential contact area, between the neutral fibers of the two adjacent pins concerned.

This connecting element, based on a bar finite element with only two nodes, takes into account:

- the gap and the contact between a wire and the neighbor pin, or between a wire and the tube,
- the thermal expansion and irradiation swelling of the pin and wire cross sections,
- the radius increase by creeping under internal pressure loading,
- the elastic rigidity of the pin crushing (locally pinching a short section of a tube),
- the contact force, altered by the two creeping behaviors, when pinching this 3D tube section,
- the counter ovalisation force due to internal pressure,
- the hot point stress and strain tensors on the internal skin (strain concentration),
- the hot point damage accumulation due to thermal creeping only.

The two main difficulties are the following ones:

- the damage evaluation supposes a really high precision in the stress tensor calculation, around 1%,
- several nonlinear structural effects should be considered in only one integration point.

These bar connecting element have been implemented in the Cast3M finite element code (http://www-cast3m.cea.fr). In the end, the bar element is validated by detailed 3D solid calculation. Then, a few relevant bundle experiments, which were conducted in the PHENIX French SFR, are fully simulated. They show good agreement to the measurements and a CPU performance gain between 100 and 1000.

**Keywords:** viscoplastic behavior, stress concentration, bar element, multiscale model, large deformation, large displacements, structural behavior, Cast3M.