FUEL ASSEMBLIES MECHANICAL BEHAVIOUR IMPROVEMENTS BASED ON DESIGN CHANGES AND LOADING PATTERNS COMPUTATIONAL ANALYSES

J. MARIN, M. AULLO, E.GUTIERREZ

ENUSA INDUSTRIAS AVANZADAS S.A. Technology and Quality. C/ Santiago Rusiñol 12; 28040 Madrid, Spain

Keywords: Bow, Computational Analyses, Mechanical

INTRODUCTION.

In the past few years, incomplete RCCA insertion events (IRI) have been taking place at some nuclear plants. Large guide thimble distortion caused by high compressive loads together with the irradiation induced material creep and growth, is considered as the primary cause of those events. This disturbing phenomom is worsened when some fuel assemblies are deformed to the extent that they push the neighbouring fuel assemblies and the distortion is transmitted along the core.

In order to better understand this mechanism, ENUSA has developed a methodology based on finite element core modelization to enable assessments on the propensity of a given core loading pattern to propagate the distortion along the core. At the same time, the core loading pattern could be decided interacting with nuclear design to obtain the optimum response under both, nuclear and mechanical point of views, with the objective of progressively attenuating the core distortion.

FUEL ASSEMBLY DESING EVOLUTION

Since the first IRI issues were detected, several fuel design changes have been introduced by ENUSA in order that make it more robust. These changes include:

- Holddown spring force reduction, so that axial stresses are reduced and fuel assembly distortion is lower.
- Inclusion of a protective grid, that provides a higher fuel axial stiffness and reduces the compression load on the guide thimbles. It is also reduced bottom span length, so that dashpot deformations are reduced.
- Introduction of a thicker thimble tube and dashpot design change. This new features increase fuel assembly lateral stiffnes.

MECHANICAL MODELS

In order to estimate FA assemblies bow inside the core, and its evolution in time, a mechanical finite elements model has been developed in ENUSA.

The Core Model is a bidimensional finite element model that consists in a row of fuel assemblies with gap elements between them and between the outermost fuel assemblies and the barrel. The operating conditions introduced to the model are temperature, fluence, hydraulic lift forces, and buoyancy forces. Under these operating conditions, the fuel assemblies distort and interact among them through the gap elements located at each grid level. Several time steps are used in the calculation with enough iterations to reach convergence. The outputs of this model are the fuel assemblies bows at the required time, and the bows incore and outcore. These last ones are obtained by removing the gaps between the fuel assemblies and the core barrel and the holddown springs, together with all loads, except the own height.

ENUSA has developed a finite element code, SAVAN, to solve the model. As the mechanical model is bidimensional, it can be studied the behaviour of only one row or column with each execution of SAVAN. Thus, to study all the core, SAVAN must be executed as many times as row/columns the core has. For each execution, it must be defined a particular input to the code, and the results must be postprocessed. A methodology has been developed to automate all this study.

SAVAN Code

SAVAN code is a 2D finite element code developed in C++, which purpose is the analysis of the evolution of the core models during the cycle. The finite elements that can be defined in the code are *non linear beams* capable to capture pre-buckling behaviour and that include creep and swelling, *linear springs*, *contacts* with friction factor depending on fluence and *gaps*. It is possible to define loads and displacement and rotation boundary conditions on the nodes. Materials definition include fluence and temperature dependency. These two magnitudes are defined through time and space dependent laws.

SAVAN code was validated by comparing the results obtained with it with those of a commercial finite element code (ANSYS). Two aspects were considered: numerical validation and creep and growth cappabilities. With respect to *numerical validation*, small tests to verify good behaviour of finite elements (beams, contacts, springs and gaps), the boundary conditions and the time dependant forces were run. Results for a complex structure integrating all the elements, and for a fuel assembly model under axial and lateral loads were also validated. In relation with *creep and growth cappabilities* validation, small tests to verifiy creep and swelling laws applied on different elements (beams and contacts) were run. A model consisting of several fuel assemblies interacting among them was also validated with ANSYS, obtaining a very good agreement between both codes.

The Core Mechanical Model

Mechanical Description of the Fuel Assembly.

The Fuel Assembly Mechanical Model is the basis of the Core Model. The finite element representation of the FA consists in:

- The guide thimble tubes, including the dashpots and the inserts (if any), are represented with beam elements with creep and growth capabilities, distributed in a way that is calculated in order to preserve the lateral flexural rigidity of all the structure.
- The fuel rods are represented by beam elements with swelling capabilities.
- Contact elements with friction are used to represent grids dimples and springs. Fuel rods are attached to the thimble tubes by this type of elements, that can be preloaded. Friction forces may relax as a function of fluence.
- Grids are modelled with beam elemets rigidly attached to the thimble tubes.
- Holddown springs are modelled with spring elements.
- To simulate in-reactor support conditions, the upper and lower nodes of the model are constrained.

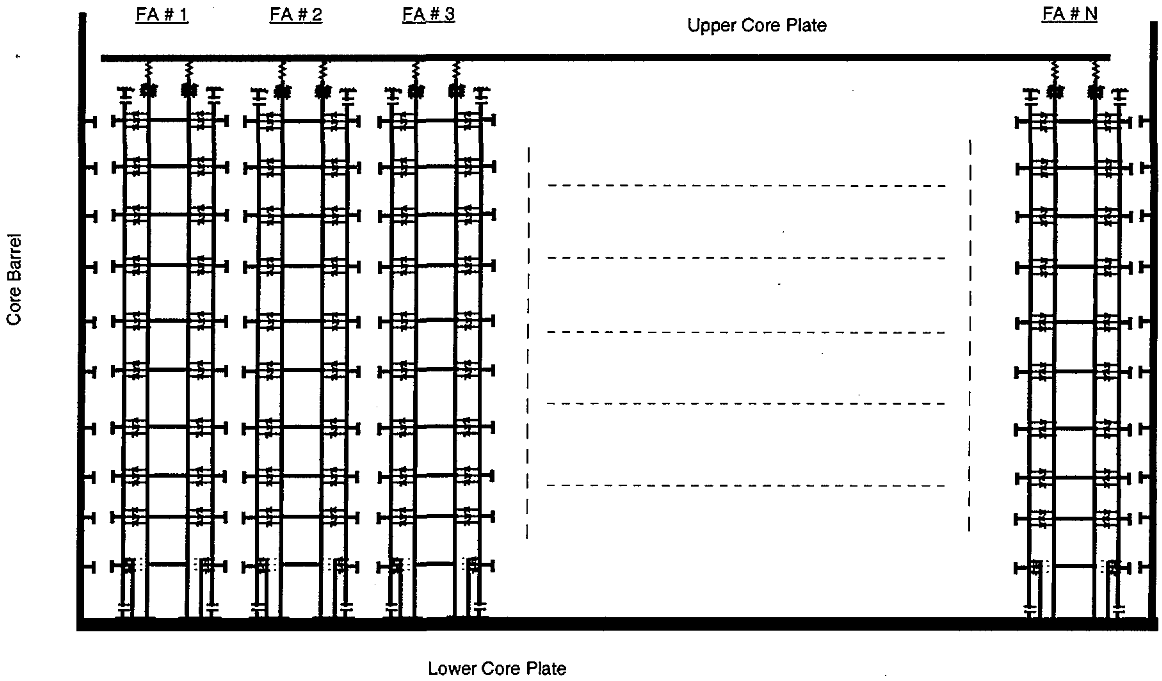
In connection with material definitions, the most important features that are considered in the model are:

- Young modulus is temperature dependent for all the materials.
- Guide thimble tube growth laws that take into account oxide formation and stress free irradiation effects that depend on temperature, fluence and fast flux are considered.
- Fuel rod growth laws that depend on fluence and fast flux are considered.
- Thimble creep rate, which is proportional to stress and fast flux, is defined.
- Grid spring relax according to a fluence dependent exponential law is implemented.

The fuel assembly models have been verified againts fuel assembly test results in air and at room temperature. The purpose of the fuel assembly lateral stiffness tests was to determine the lateral load deflection characteristic curves of an axially preloaded fuel assembly . The fuel assemblies were laterally loaded at grid level-5 and the lateral displacements of each grid was recorded by two displacement indicators to be able to cancel any rotation of the assembly due to eccentric loading. The load versus deflection characteristics were non-linear due to the fuel rods slipping and dimple lift-off in the grids when the static drag force is overcome. This results in a reduction in lateral stiffness at higher deflections. Analysis with SAVAN code of the finite element fuel assembly model were done in order to adjust variables as grid springs and dimples drag forces or the coefficient of friction in the contacts.

The Core Model

The core model consists in a row of fuel assembly models with gaps between them and between the outermost fuel assembly and the core barrel. An squematic representation is shown in the figure.



Core Mechanical Model

The main actions considered in the model are:

- *Buoyancy and hydraulic forces*, distributed as vertical nodal loads in the thimble tubes nodes at grid elevations.
- *Fuel assembly weights*, distributed in the same nodes as hydraulic loads.
- *Holddown spring forces.* In the initial state, these forces are represented as nodal loads applied on the thimble tubes upper nodes. Under operation, they are represented by the action of a precompressed spring element.

Temperature and fluence data at spans and grids at different states of the cycle is provided by nuclear design. Least squares aproximations are done to estimate the value of these magnitudes at any step by the definition of parabolic functions of fluence and temperature vs. time.

Cycle Analysis.

As SAVAN is a two dimensional code, it is not possible to obtain the cycle results in a single run and the analysis must be divided into rows and columns. For each row and column, the time history analyisis considers three main stages:

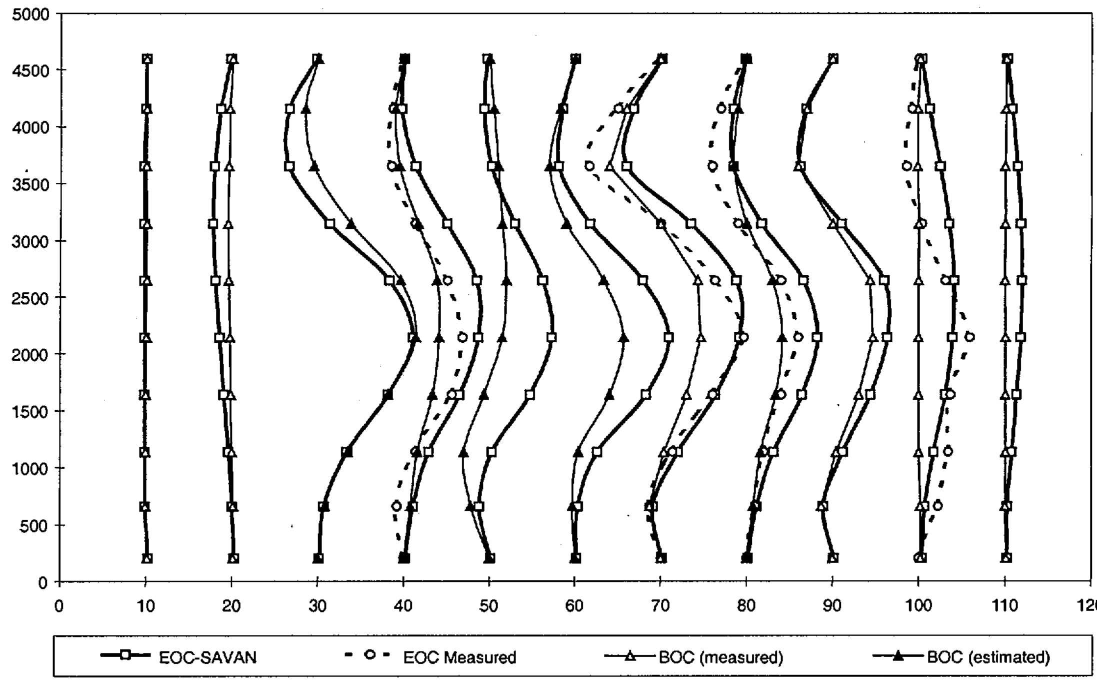
1) *Initial state*, at room temperature, In this state, the fuel assemblies loading inside the core is simulated. As the fuel assemblies are boxed up with a very small gap, the fuel assemblies initial bow change. Fuel assemblies weights and holddown springs cold forces are considered.

2) *Operation at hot conditions*. The cycle effective time is simulated. The considered operating conditions take into account buoyancy and hydraulic forces, holddown springs hot forces, and neutronic flux that cause grid forces relaxation, fuel assemblies growth and creep deformation.

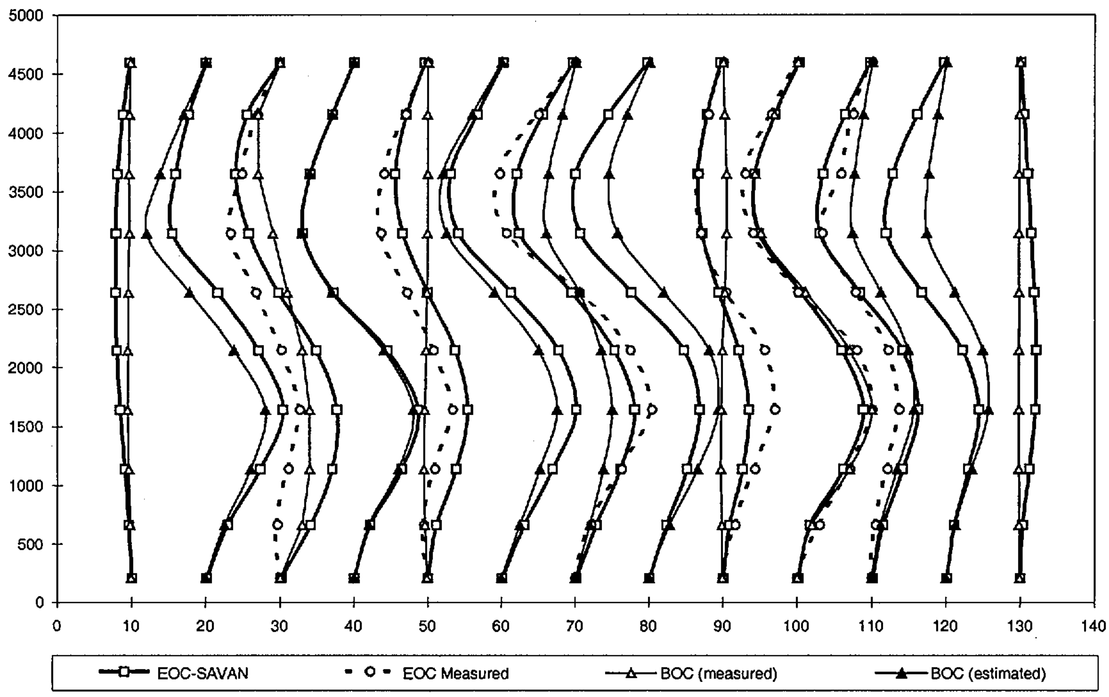
3) *Final state*. Temperature decreases to room temperature. Holddown springs, buoyancy and hydraulic forces and inter-assemblies constraints are removed. All the elastic deformations dissapear, and comparison between fuel assemblies bows estimations and plant measurements, that are done in the pool, may be carried out.

Model validation

A verification of the appropriateness of the code was been done using fuel assemblies bows measurements in several cycles in an operating plant. As an example, the next figures compare the estimated bow with the measured one for one row and one column of the core. In these figures, the distance between adjacent fuel assemblies is not the distance in the core, but it has been fixed in ten milimeters to facilitate the fuel assembly bow representation. Not all the initial fuel assembly deformations were known for these analyses and some of them were estimated based on known fuel assemblies shapes of the neighbour fuel assemblies in previous cycles. These cases are indicated in the graphs. It must be noted than initial bow is a critical magnitude, because the most important effect in relation with this problem is its propagation and magnification along the core during the operation.



Bow estimation (I)



Bow estimation (II)

MECHANICAL EVALUATION OF LOADING PATTERNS

In order to minimize bow apparition and magnification effects during the operation, mechanical aspects must be considered when loading patterns are defined and evaluated. ENUSA methodology in relation with this aspect includes two estrategies:

1) Mechanical guidelines setting-up in relation to the position and distribution of the fuel assemblies inside the core depending on its initial bow and burnup. Basically, these guidelines define the cases that are considered more dangerous in relation to bow propagation and IRI risk, and restrict the possibility of their appearance.
2) Loading patterns mechanical analysis with SAVAN code so that fuel assemblies bows at the end of the cycle are predicted. Computational procedures have been implemented to automate the definition of the input files to the code (one for each row/column), the execution of SAVAN, and the postprocessing of the results. As a result, the duration of the mechanical evaluation does not suppose a significant delay in the overall of the engineering work.

As an example of the results of the application of this methodology, in the next figure, the evolution of the RCCA insertion times (that are directly related to fuel assemblies distortions) along different cycles of a nuclear plant is shown. As it can be seen, the reduction is very significant.