FINITE ELEMENT PELLET-CLAD INTERACTION MODEL

A. ALUJEVIC, P. ŠKERGET
Institut Jozef Stefan, University of Ljubljana,
POB 199, 61001 Ljubljana, Yugoslavia

B. CERNEJ
College of Technology and Engineering, University of Maribor,
Smetanova 17, 62000 Maribor, Yugoslavia

ABSTRACT

During the nuclear reactor operation, one of the most important limiting factors in the fuel element duty is the mechanical interaction of fuel pellets and cladding tubes. This interaction produces cyclic stresses and strains in the cladding, and these in turn consume clad fatigue life.

Computer analyses of the behaviour of fuel pins have shown that the deformation of its components have significant effect on its temperature distribution, resulting mainly from variations with time of the gaps between the fuel pellet and cladding tubes. Changes in temperature distribution in turn influence the deformations of the pin and the stresses developed in its components. Therefore the heat conduction calculations (with convection heat transfer coefficient as a boundary condition) and the stress-strain calculations cannot be made separately, but must be made simultaneously. To this end, existing structural analysis body of revolution finite element code BREL and thermal analysis finite element code TEMPEL have been combined in order to yield a new TEMPEST computer programme.

This code is using thoroidal finite elements, with simplex linear triangulation on the meridional cross-section of the fuel element. The simultaneous computation is performed by solving thermal and structural equations, $\dot{u} = g$ and $K u = f$ respectively, inside of each individual time step. With respect to the thermal conduction analysis, Dirichlet's, Neumann's and mixed type (of the third kind) boundary conditions are considered (while neglecting Stefan's law due to the nonlinearity), and within the structural analysis, elastic Hooke's generalised law is used with allowance being made for thermal expansion, irradiation induced shrinkage or swelling, creep, cracking etc. In due course, our code is to be implemented by circumferential Fourier's analysis for azimuthally dependent loading of geometrically axisymmetric bodies.

Some test runs with TEMPEST code produced satisfying results. In this paper, hoop stress distributions are presented in form of isobars (isopascals) on the meridional cross-section of an UO$_2$ fuel pin with metal cladding material, for various conditions of vanishing gaps between the pellet and tube.
1. INTRODUCTION

In nuclear reactors mechanical interactions may frequently occur between the fuel pellet and its cladding. This phenomenon has been studied in detail for several years by various authors, among others by Head [1] and Rashid [2]. There is an additional cause of the potential interaction failure - the combined effect of differential thermal expansion driven localised stresses and aggressive fission products, primarily iodine. The chemical aspect of pitting corrosion problem is not going to be a subject of this paper. The study is to be limited to a pure mechanical interaction of the fuel with external cladding.

2. FINITE ELEMENTS FOR TEMPERATURE AND STRESS ANALYSIS

Few years ago, a computer code HASSAN [3] has been written for simultaneous temperature and thermal stress analysis of nuclear reactor fuel elements, with an integration routine for radial variation of unknowns at several axial positions along the cooling channel. Having gained a certain amount of experience with the above code, and also having run two individual finite element codes in two dimensions (radial-axial), i.e. TEMPEL [4] for temperature distribution and BREL [5] for thermal stress analysis, present authors came to the conclusion to synthesise these codes with an aim to produce a joint TEMPEST [6] programme, its name being an acronym for Temperature & Stress computing code. The two FEM techniques represent a well known procedure, each yielding a system of linear algebraic equations to be solved by numerical means (H u = f and K u = f). Stiffness and conductivity matrices, and RHS column vectors (forces, fluxes) are known, while LHS vectors (displacements, temperatures) appear to be unknown variables at the nodal points of the finite element mesh. Details on the formation and solution techniques can be found elsewhere in the literature on the subject. It can be noted, that our codes (TEMPEL, BREL, TEMPEST) use simplex linear triangular finite elements on a meridional cross-section of a body of revolution. The method of solution used is the iterative Gauss-Seidel technique. Alternative methods can also be applied, e.g. Gaussian elimination, Cholesky's decomposition etc., with frontal option being a very versatile tool.

A special care has to be given to the appropriate boundary conditions, thermal and mechanical. At some points of the cross-section temperatures may be prescribed, and the conductivity matrix has to be adapted by eliminating rows and columns. Another option is a magnification of diagonal terms by a very large number (1.E20). Additional simple boundary conditions dealt with are adiabatic conditions ("unloaded" case) where the matrix remains unchanged. If heat flux is prescribed on individual boundaries or exchange coefficient to the environment is given, these can all be dealt with by the condition in the form of a·t_i+b·t_o+c·t+d (where BC 1st kind are Dirichlet's, 2nd kind Neumann's, 3rd kind of the mixed type). At the moment, 4th order thermal boundary condition cannot be taken into account (radiation Stefan's law) due to its nonlinearity. In respect to the mechanical boundary conditions, fixed, restricted and elastically supported nodes are incorporated by a modification of the flexibility matrix.
Material properties in the structural part of the TLMPEST code can be taken for individual elements, and the analysis is limited to the elastic case, with a generalised Hooke's law for nonisotropic materials. An allowance has been made for thermal expansion strain, irradiation induced shrinkage and swelling, creep (isochoric and nonisochoric), noncircumferential cracking etc. The code is also being to be adapted for plastic behaviour of the cladding.

For the circumferentially dependent analysis (loading, cracking), the Fourier harmonic decomposition is under consideration to be incorporated in the existing FEM code (ref. Wilson /7/). Temperatures, loads, displacements, strains and stresses may be considered to be \( X(r,z,\phi) = \sum \frac{X_n(r,z) \cdot \sin(n\phi)}{\cos} \), \( n=0,N \). Thus, \( N+1 \) independent systems of equations for \( M \) elements are to be solved, and the whole procedure tends to be very slow with a poor convergence rate. At the moment, this part of our research has not produced satisfactory results yet.

3. MECHANICAL INTERACTION MODELS

Only some simulation models by finite elements for the pellet-cladding interaction are known to be worked out in the practice, among these one developed by Kaegstra /8/ and another by Norman /9/. The first of these models can be characterised as a nodal mis-matching technique with an allowance for the interface friction, using an iterative method for determination of bending moments needed to produce a plane strain condition as a realistic boundary condition in the cladding tube part, cut from the whole stack. The second model is using an additional ballast number of finite elements on the top as a cover part simulating realistic boundary conditions in a different way.

In the work, presented in this paper, a simpler model is used with nodal point matching technique (Fig. 1). Boundary conditions at the upper cut of the cladding tube are proposed to be with elastic support, allowing free radial movements. Nodes at the lower cut of the tube and of the pellet are restricted to radial movement only, as usual in the finite element practice.

4. TEST CASE COMPUTATION

Fuel pellet with an O.D. of 10 mm has been assumed with and without an internal hole of an I.D. of 2 mm. The fuel pin is cladded by an idealised metal sleeve with I.D. of 10.2 mm and O.D. of 12.2 mm. For the test case, only a single pellet from the whole fuel element stack, with an appropriate slice of the cladding tube, has been considered while analysing the performance of the selected model.

Thermal load has been accounted for with 500 degrees Centigrade in the heat generating zone (1500 °C being the central line maximum temperature), and 50 degrees Centigrade in the heat conducting part (900 °C being the outer wall temperature minimum level value). The distribution of the temperature in the pellet is known to be parabolic, while in the cladding tube a linear distribution has been considered, thus simulating the logarithmic one by an adequate
simplification. Axial temperature drop has been eliminated by adiabatic boundary conditions at the top of the fuel pellet.

Material properties of the ceramic fuel and metal cladding have been taken as follows:

<table>
<thead>
<tr>
<th>Fuel:</th>
<th>Young’s modulus 163 GPa</th>
<th>Poisson’s ratio 0.2</th>
</tr>
</thead>
<tbody>
<tr>
<td>Cladding:</td>
<td>Young’s modulus 206 GPa</td>
<td>Poisson’s ratio 0.3</td>
</tr>
<tr>
<td></td>
<td>C.T.E. 6 ppm/K</td>
<td>C.T.E. 5 ppm/K</td>
</tr>
</tbody>
</table>

In the finite element mesh at the initialisation stage, prior the touching of the two parts, there are 50 finite elements with 42 nodes in the pellet, and 24 elements with 21 nodes in the cladding. Both meshes are considered to be relatively coarse and may be refined for a more detailed stress analysis.

When the interaction takes place during the reactor operation, the total of 63 nodes reduces gradually down to 56, when the complete gap closes by an ideal contact. This yields 8 options in our idealisation.

16 consecutive runs on the CYCLR 72 computer at the RRC in Ljubljana (terminal in Maribor) produced results, a selection of which is given on Fig. 2 for the hollow, and on Fig. 3 for the solid pellet. These results are presented on a meridional cross-section in form of isobars ("isopascals") measured in kN/mm² (100 MPa) for the hoop stress component. Radial, axial and shear stresses could also be plotted, values of them being lower as the circumferential stresses.

From these figures, the following stress behaviour could be evaluated: Initially, there are hoop stresses of approximately ±300 MPa in the pellet, and relatively low values in the cladding. By positive (+) tensile, and by negative (-) compressive stresses are signed. During the interaction, values have climbed up to +700 MPa in the tube, and down to -600 MPa at the pellet centre, while maximum tensile stress in the pellet occurs to be +300 MPa in top corner. Tensile limit strength of stainless-steel or zircalloy (UTS = 30 t/sqinch), and for the uranium dioxide in the reactor core environment appear to be higher, but compressive stresses indicated at the I.D. (or C.L. respectively) in the ceramic body may produce some cracking, usually present in UO₂ pellets, resulting also due to various additional reasons during the operation of the reactor core (ref. Guha et al. /10/).

5. CONCLUDING REMARKS

The lifetime of fuel elements has to be analysed with respect to the power history and its direct effects on thermal stress variation. Startups, ramping and load following behaviour (ref. /11/) with fresh fuel elements are considered to represent no major problem with respect to potential PCI failures due to chemical and mechanical causes. On the other hand, with irradiated fuel rods vendors have determined the failure-nonfailure threshold for very fast power ramps of about 3 to 5 %/hr in PWR and BWR fuel. The FEM model described in this paper can be useful for this kind of analysis (mechanical point of view), including creep, cracking and other important irradiation induced
effects in the $\text{UO}_2$ and metal cladding materials.

6. REFERENCES


/5/ ALUJEVIć, A., HEAD, J. L., "Fuel element stresses at discontinuities and interactions by finite elements in two dimensions", Atomkernenergie 21, 75-80 (1973)


/11/ KTG/ENS/JRC Meeting on Ramping and Load Following Behaviour of Reactor Fuel, Petten, Holland, November 30 – December 1, 1978 (Proceedings to be published as an EUR report)

Acknowledgements

Authors gratefully acknowledge the support by the Slovenian Research Council for the work described in this paper. Thanks are also due to Messrs. J. Head, P. Keegstra, B. Eysink, A. Jezernik, M. Copić, L. Prelog, B. Vrečko, and H. Herman for valuable help and discussions on the subject of this paper.
Fig. 1 Pellet-Clad Interaction Model by Finite Elements in 2-D
Fig. 2 Hoop stress results for the hollow pin (hN/mm² = 100 MPa)
Fig. 3 Hoop stress results for the solid pin (hN/mm²=100 MPa)