A Finite Element Fuel Cladding Model with an Interactive-Mixed Formulation for Creep Analysis

J. Matheson

Babcock & Wilcox, P.O. Box 1260, Lynchburg, Virginia 24505, U.S.A.

L. Mansfield, E. Sieveka

University of Virginia, School of Engineering and Applied Science, Charlottesville, Virginia 22903, U.S.A.

Abstract

Babcock & Wilcox has recently undertaken development of two new computer codes for the analysis of the thermal and mechanical behavior of light water reactor fuel rods. Early work in this program, in conjunction with the Univeristy of Virginia, has resulted in a two-dimensional finite element model for the fuel rod cladding. In the elastic regime a classical displacement based solution scheme has been successfully benchmarked. To solve the creep problem the strain rates (velocities) as well as the displacements are approximated with finite element basis functions. The resulting system of non-linear equations is solved employing Newton's method in conjunction with a variational iteration scheme. Time integration is handled in a novel way by applying the backward Euler Method directly to the stresses rather than indirectly through the creep strains.

1. Introduction

For the past two years, Babcock & Wilcox in conjunction with the University of Virginia has been developing new computer methods to accurately describe the behavior of fuel rods in a nuclear reactor. One of the products of this effort is the FUMAC-84 code which is designed to predict the mechanical performance of the fuel - especially the stress state of the cladding. Because the focus of FUMAC-84 is the cladding rather than the pellet, early efforts in code development have sought to derive a detailed yet efficient model of the cladding. After a careful consideration of the complex boundary conditions and the non-linear nature of the problem, a finite element representation was chosen.

Use of commonly available finite element codes was considered. However, the demands for complete solution of this problem are specialized. The code of choice must be capable of modeling an anisotropic material for both elastic and viscoplastic response to changing thermal and mechanical boundary conditions. A capability to "customize" material models is required in order to include the effects of temperature and irradiation. Beyond this, some "transparency" of the coding is required so that solution methods could be optimized during rapid creep which has been described by Nuno et.al.[1], as inherently unstable. All of these demands could not be met by any code considered so it was decided to develop a highly specialized fuel rod code (FUMAC-84) which has some unique features. Figure 1 illustrates the logic flow for the code.

The FUMAC-84 code is portable - written in ANSI standard FORTRAN77. All necessary routines for matrix manipulation and solution are included so that no special mathematical library calls are required. The geometry description is specialized to fuel rods so that the physical model is quickly defined by node generation techniques thus eliminating the need for node/element incidence tables. Design-specific models for anisotropic elastic and viscoplastic response as well as irradiation damage are allowed. Finally, an implicitly stable solution scheme is employed for the solution is the creep problem.

The purpose of this paper is to present some background on the general linear finite element model being employed for FUMAC-84 and to explain in detail the unique solution technique which has been derived for the non-linear part of the solution. First, a short description of the physical problem being modelled is in order. The case of interest for the failure analyses of fuel is when the pellet is in hard contact with the cladding. Thus, the modelling of both pellet and cladding is required.The extreme complexity of the fuel pellet behavior recommends an empirical approach. The cladding, on the other hand, can be realistically dealt with using finite elements. Ring elements which have seen large use due to computational simplicity, fail to describe the problem completely. Higher order finite elements provide more detailed interpretations of stress and strain. The appropriate use of two-dimension finite elements can solve both the lateral ridging problem as well as predict azimuthal stress concentrations which may initiate Stress Corrosion Cracking (SCC). In our approach the behavior of the pellet is basically considered to be an interactive boundary condition on the cladding. The solution proceeds iteratively in a quasi-static manner in time.

2. The Linear Model

Modeling of the fuel rod cladding is currently being approached through the use of 2D finite elements in both axisymmetric and plane stress/strain geometries. It is expected that a combination of the information obtained from each of these model geometries will adequately predict the behavior of the cladding. If in fact 3D modeling becomes necessary, it may be possible to use the 2D calculation to predict when full 3D modeling should begin, thereby speeding program execution during initial time steps.

Either 4 or 9 node isoparametric elements can be chosen to span the cladding for both of the above geometries. In the case of plane stress/strain, 9 node elements with their quadratic shape functions are recommended to better represent the curved boundaries encountered (see Figure 2).

Solving for stress, strain, and displacement in the cladding is begun by obtaining an elastic solution for either of the above geometries, given an initial loading. The physical model is that of linear elasticity. It is assumed that cladding displacement will be described by the following equation:

$$[K]\ u = F \qquad \text{eq. (1)}$$

where [K] represents the elastic stiffness matrix, u is a vector of displacements at each node and F is a vector of loads associated with each node. Solution of the above equation for u gives the condition of the cladding in elastic equilibrium. From u it is possible to obtain the stress and strain fields from the equations below.

$$\text{Strain} = [D]\ u \qquad \text{eq. (2)}$$

$$\text{Stress} = [E]\ \text{Strain} \qquad \text{eq. (3)}$$

[D] represents a matrix of derivatives which relate displacement to strain. For plane stress/strain [D] is given by:

$$[D] = \begin{vmatrix} \partial/\partial x & 0 \\ 0 & \partial/\partial y \\ \partial/\partial y & \partial/\partial x \end{vmatrix} \qquad \text{eq. (4)}$$

and for an axisymmetric model by:

$$[D] = \begin{vmatrix} \partial/\partial r & 0 \\ 0 & \partial/\partial z \\ 1/r & 0 \\ \partial/\partial z & \partial/\partial r \end{vmatrix} \qquad \text{eq. (5)}$$

[E] is the material properties matrix with terms defined by the Young's modulus, shear modulus, and Poisson's ratio for the cladding material.

In order to verify the correctness of the code for the elastic solution, it was benchmarked against the analytical solution for an internally pressurized cylinder with end boundaries fixed in the axial direction and assuming material isotropy. For such a cylinder, the geometry is axisymmetric and displacements are independent of z, hence a plane strain situation. The dependence of the radial displacement on r, for plane strain, is given by:

$$u_r = \frac{P}{E}\left(\frac{a^2}{b^2 - a^2}\right)(1 + \nu)\left[(1 - 2\nu)r + \frac{b^2}{r}\right] \qquad \text{eq. (6)}$$

Here, P is the pressure, E is the Young's modulus, ν is the Poisson's ratio, a is the inner radius of the cylinder, and b is the outer radius. Using the following parameters:

a = .50 in

b = .55 in

P = 1500 psi

E = 1.1254E+7 psi

and ν = .3478,

the results shown in Table 1 were obtained. The numerical displacement solution was indeed independent of z and the radial displacements agreed extremely well with the analytical values (see Table 1).

In the numerical analysis of cladding creep, the elastic solution provides the initial conditions needed to begin the subsequent iteration over time. In addition, the elastic stiffness matrix, which is also needed in the creep solution, is retained and need not be recalculated as long as the material properties remain constant. In the next section we describe the time iteration algorithm used to solve for creep.

3. Modeling Creep Behavior

A basic assumption in modeling creep behavior is that the total strain can be expressed as the sum of the elastic, creep, and thermal strains (Note: irradiation growth strains are treated in a similar fashion.)

$$\varepsilon_{ij} = \varepsilon_{ij}^{e} + \varepsilon_{ij}^{c} + \varepsilon_{ij}^{TH}. \qquad \text{eq. (7)}$$

The creep strain rate can be related to the stress using a modified equation-of-state approach. We use

$$\dot{\varepsilon}_{ij}^{c} = \gamma S_{ij}, \qquad \text{eq. (8)}$$

where S_{ij} is the component of the deviatoric stress tensor, which for isotropic bodies is given by

$$S_{ij} = \sigma_{ij} - \frac{1}{3} \sigma_{mm} \delta_{ij}, \qquad \text{eq. (9)}$$

and where

$$\gamma = \frac{3}{2} \frac{\dot{\bar{\varepsilon}}^c}{\bar{\sigma}} \qquad \text{eq. (10)}$$

where $\dot{\bar{\varepsilon}}^c = f(\bar{\sigma}, T, t)$ effective creep strain rate and $\bar{\sigma}$ is the Von Mises effective stress given by

$$\bar{\sigma} = (\frac{3}{2} S_{ij} S_{ij})^{\frac{1}{2}} \qquad \text{eq. (11)}$$

The creep strains can also be represented by

$$\dot{\varepsilon}^C = \gamma [S] \sigma \qquad \text{eq. (12)}$$

where [S] is a matrix of coefficients obtained from eq. (9).

The elastic strains are related to the stress by

$$\sigma = [E] \varepsilon^e = [E](\varepsilon - \varepsilon^C - \varepsilon^{TH}) \qquad \text{eq. (13)}$$

and the thermal strains are computed from

$$\varepsilon^{TH} = \alpha_m (\theta - \theta_R)\delta, \qquad \text{eq. (14)}$$

where $\delta = (1, 1, 1, 0, 0, 0)$ and θ and θ_R represent temperatures.

If (13) is differentiated with respect to time, one obtains

$$\dot{\sigma} = [E](\dot{\varepsilon} - \dot{\varepsilon}^C - \dot{\varepsilon}^{TH}) = [E]([D]v - \gamma[S]\sigma - \dot{\varepsilon}^{TH}) \qquad \text{eq. (15)}$$

where v are the velocities and [D] is the operator matrix incorporating the strain-displacement relations. Equation (14) can be integrated in time using either the backward Euler method or the trapezoidal rule. Below, for simplicity in exposition, we use the backward Euler method. Suppose the displacements and velocities are discretized by

$$u = \sum_i N_i U_i, \quad v = \sum_i N_i V_i \qquad \text{eq. (16)}$$

where the N_i are basis functions and the U_i and V_i are nodal values. Then the equilibrium equations may be expressed by

$$\int_\Omega [B]^T \sigma \, d\Omega = R \qquad \text{eq. (17)}$$

where $[B] = [D]N$ and

$$R = \int_\Omega N^T f \, d\Omega + \int_{\Gamma_\sigma} N^T \hat{\sigma} d\Gamma. \qquad \text{eq. (18)}$$

where the following nomenclature applies:

Ω is the solution domain,

Γ is the boundary of the solution domain, and

f and $\hat{\sigma}$ are the body and traction forces, respectively.

At each time step, our algorithm can be expressed as

1. Set ${}^{t+\Delta t}\sigma^{(o)} = {}^{t}\sigma$, ${}^{t+\Delta t}V^{(o)} = {}^{t}V$

2. For i=0,1,2,... until $\Delta V \to 0$

   a. At each stress determination point, solve ${}^{t+\Delta t}\sigma^{(i+1)} = {}^{t}\sigma$ $+ \Delta t[E](B^{T}\ {}^{t+\Delta t}V^{(i)} - {}^{t+\Delta t}\gamma^{(i+1)}\ [S]\ {}^{t+\Delta t}\sigma^{(i+1)} - {}^{t+\Delta t}\varepsilon^{TH})$ for ${}^{t+\Delta t}\sigma^{(i+1)}$ by Newton's method

   b. Solve

   $$K^{t+\Delta t}\ \Delta V^{(i+1)} = \frac{1}{\Delta t}\ ({}^{t+\Delta t}R - \int_{\Omega} B^{T}\ {}^{t+\Delta t}\sigma^{(i+1)} d\Omega)$$

   for ${}^{t+\Delta t}\Delta V^{(i+1)}$ where $[K] = [B]^{T}[E][B]$ is the elastic stiffness matrix.

   c. Set ${}^{t+\Delta t}V^{(i+1)} = {}^{t+\Delta t}V^{(i)} + {}^{t+\Delta t}\Delta V^{(i+1)}$

The thermal strain rate ${}^{t+\Delta t}\dot{\varepsilon}^{TH}$ can be computed from

$$ {}^{t+\Delta}\dot{\varepsilon}^{TH} = \frac{1}{\Delta t}\ [{}^{t+\Delta t}\varepsilon^{TH} - {}^{t}\varepsilon^{TH}] \qquad \text{eq. (19)}$$

where ${}^{t+\Delta t}\varepsilon^{TH}$ and ${}^{t}\varepsilon^{TH}$ are determined from eq. (13). The total strain can be calculated anytime it is desired from

$$ {}^{t+\Delta t}\varepsilon = [B]^{t+\Delta t}U, \qquad \text{eq. (20)}$$

where

$$ {}^{t+\Delta t}U = {}^{t}U = \frac{\Delta t}{2}\ ({}^{t+\Delta t}V + {}^{t}V). \qquad \text{eq. (21)}$$

Our algorithm is contained in [2]. It differs from previous techniques to solve creep problems in that the creep strains do not need to be explicitly solved for at each time step. Experimental results presented in [2] have shown it to be up to twice as fast over many time steps for the same accuracy as the algorithm used in ADINA [3].

4. Conclusions

In the course of solving the problem of creep in fuel rods, a new solution algorithm has been devised. This method promises to be faster and more stable than previous approaches. The cladding model described is part of a larger effort to model fuel mechanical behavior in a simple yet accurate fashion. The basic approach is to focus on the cladding as the primary constituent of the problem and treat the pellet thermal and mechanical behavior as dynamic boundary conditions. Future work will be directed at interfacing these boundary conditions with the cladding model.

5. References

[1] H. Nuno, S. Ogawa, T. Sato, "On The Treatment of Numerical Problems Due to Rapid Creep Rate in the Fuel Performance Calculation by FEM," (Mitsubishi Atomic Power) Presented at the 7th IWGFPT (1980).

[2] C.W. Heaps and L. Mansfield, "An Improved Solution Procedure for Creep Problems," (submitted for publication).

[3] ADINA Engineering Inc. Report AE 81-1, "ADINA - A Finite Element Program For Dynamic Incremental NM Linear Analysis," ADINA Engineering Inc., Watertown, Mass., 1981.

Table I

Results of Linear Analysis Benchmark

$U_r (x\ 10^{-4}\ in)$

r (in)	Analytical	Numerical Linear Elements**	Numerical Quadratic Elements**
.5000	6.477	6.476	6.477
.5125	6.384	6.383	6.384
.5250	6.296	6.295	6.296
.5375	6.214	6.213	6.214
.5500	6.137	6.136	6.137

* 4 elements in radial direction

** 2 elements in radial direction

FIGURE 1

FUMAC-84 DESIGN LOGIC FLOW

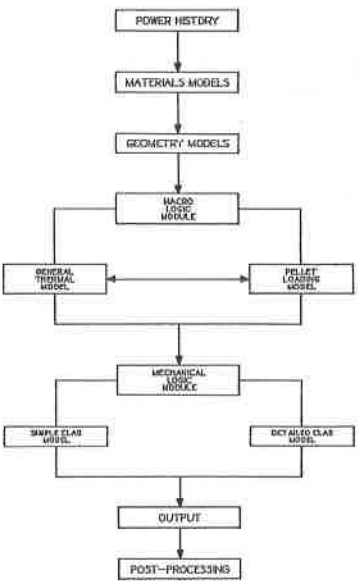
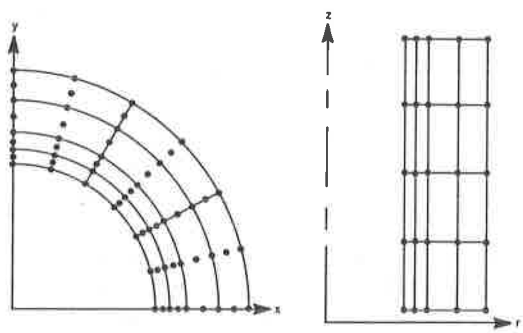


Figure 2
FUMAC-84 Elements



Plane Stress/Strain Geometry Showing 9-node Elements

Axisymmetric Geometry Showing 4-node Elements