Journal of the Korean Nuclear Society
Volume 25, Number 4, December 1993

A Study on the Fuel Assembly Stress Analysis for Seismic and Blowdown Events

Il Kon Kim

Korea Atomic Energy Research Institute

(Received September 10, 1993)

지진 및 냉각재상실사고시의 핵연료집합체 응력해석에 관한 연구

김일곤

한국원자력연구소

(1993. 9. 10 접수)

Abstract

In this study, the detailed fuel assembly stress analysis model to evaluate the structural integrity for seismic and blowdown accidents is developed. For this purpose, as the first step, the program MAIN which identifies the worst bending mode shaped fuel assembly(FA) in core model is made. And the finite element model for stress calculation of FA components is developed. In the model the fuel rods (FRs) and the guide thimbles are modelled by 3-dimensional beam elements, and the spacer grid spring is modelled by a linear and rotational spring. The constraints come from the results of the program MAIN. The stress analysis of the 16×16 type FA under arbitary seismic load is performed using the developed program and modelling technique as an example. The developed stress model is helpful for the stress calculation of FA components for seismic and blowdown loads to evaluate the structural integrity of FA.

요 약

지진 및 냉각재상실사고시 핵연료집합체의 건전성 확인은 원자로심모델의 핵연료집합체 집중질량모델을 이용하여 지지격자에 발생한 충격 해석치와 동적좌굴시험치와의 비교를 통해 사고시의 핵연료집합체 건전성을 평가하여 왔다. 그러나 이 방법은 사고시 핵연료집합체 부품별 설계요구사항 만족여부를 평가하는데 미흡하여 본 연구에서는 지진 및 냉각재상실사고시 핵연료집합체 구조적건전성 평가를 위한 수평방향 핵연료집합체 응력해석모델을 개발하였다. 이를 위해 첫번째 단계로써 원자로심모델의 해석결과인 각 절점에서의 변위와 회전각으로부터 응력을 계산하고 가장 큰 응력을 갖는 핵연료를 찾아내는 MAIN이라는 전산프로그램을 개발하였다. 그리고 다음단계로써 이 프로그램에서 구한 핵연료집합체 변위와 회전각을 이용하여 핵연료집합체의 주요부품에 가해지는 응력을 계산하기 위한 핵연료집합체 응력해석모델을 개발하였다. 이 모델은 집합체주요부품인 안내관과 연료봉을 3차원보요소로, 지지격자스프링을 선형 및 회전스프링으로 각각 모델링 하

였으며, MAIN 프로그램의 출력인 집합체의 변위를 구속조건으로 사용하였다. 또한, 개발된 프로그램과 응력해석모델을 이용하여 하나의 적용 예로써 임의의 지진하중하에서 16x16형 핵연료집합체에 대한 응력해석을 수행하였다. 이 모델을 개발하므로써 지진 및 냉각재상실사고시 핵연료집합체 설계요구사항 만족여부를 평가할 수 있는 기틀을 마련 하였다.

1. Introduction

In the design of FA, several accident conditions are postulated and analyzed to assure the safety of the system. Among these assumed accidents, the safe shutdown earthquake(SSE) and loss of coolant accident(LOCA) are the most severe ones from the point of the loads applied to the FA components. The design requirements for these condition III and IV loads[1] aim at preventing fission gas release in the environment. For the FA, the following design criteria must be fulfilled :

(a) integrity of the FA in order to allow the control rod insertion for the safe shutdown of the plant,

(b) coolability of the core : residual heat removal from the fuel.

During a seismic or blowdown event, the ground motion caused by a earthquake or the shock wave caused by a primary pipe break, is transmitted to the core plate through the primary circuit and reactor internals. The horizontal core support plate motion induces a collision of FAs . The foreign FA vendors have developed the their own detailed FA stress model to calculate component stress for seismic and blowdown events[2,3]. But in domastic case the FA integrity for the events is evaluated by comparing the calculated impact force of spacer grid in the core model with the dynamic buckling test results of spacer grid[4].

The main assumption of this evalution method is that the impact locations are limited to the spacer grids which are the widest part of the FA. So far the design criteria of FA for condition III and IV loads imply that spacer grid is not buckled dynamically.

For the evaluation of the FA design criteria for seismic and blowdown events, however, the lumping all fuel assemblies in core model as shown in Fig.1 is not accurate because reduced FA model generates unrealistic stress results. Therefore the detailed FA model is necessary to evaluate the FA design criteria during postulated pipe breaks or earthquake.

In this study the developed computer program MAIN which identifies the worst bending mode shaped FA in core model is described as the first step. And, as the next step, the detailed FA stress model, of which constraints come from the results of the program MAIN and by which the peak stress is calculated, is investigated using the finite element code KWUSTOSS-STATIC[5]. Also, as an example, the stress analysis of the 16x16 type FA is performed using the developed program and model.

2. Model Development

2.1. Development of the program MAIN

In the core model, the fuel assemblies(FAs) are modeled as uniform beam and the impact forces of each FA due to gap closure are evaluated at each time step. Therefore a lot of information is obtained by the core model analysis for seismic and blowdown events. Among these information, it is necessary to identify the worst bending mode shaped FA in stress point of view.

The computer program MAIN is made to identify the worst bending mode shaped FA in core model. The calculations to determine the 'reply' of the core model to the postulated seismic and

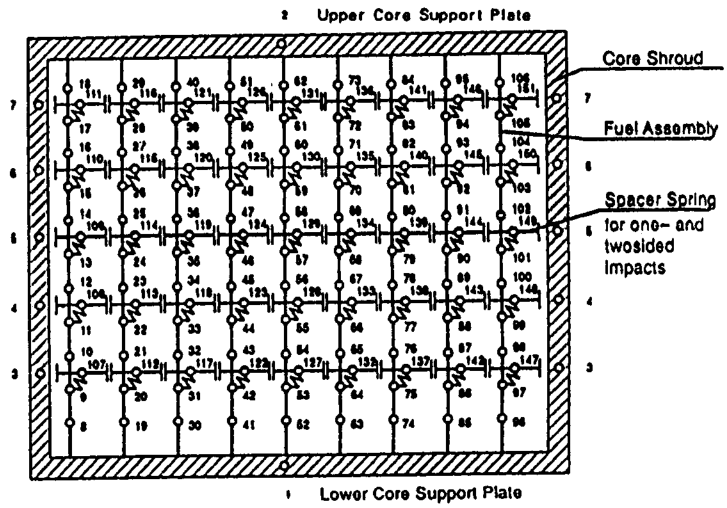

Fig. 1. Core Model(9 FA Series Model)

blowdown events are performed, and the program MAIN uses as input these data to calculate estimates for the stress at each segment of FA and at each time step. Hereby the peak stress can be identified. The exact value of the stress is calculated in the next step by applying a detailed FA stress model.

For the development of the program MAIN the following assumptions are introduced :

Core model is supposed to have :

- equal number of nodes and equal number of spacer grids for each FA.
- one node between two spacer grids.
- node 1 corresponding to the lower core support plate(LCSP).
- node 2 corresponding to the upper core support plate(UCSP).
- consecutive numbering system of FA nodes.
- one node defined between the lowest spacer grid and the lower core support plate/ between the highest spacer grid and the upper core support plate.

And for the preliminary estimates of the stress of each segment, the following assumptions are introduced :

- the rotational displacement Φ of spacer grid level are equal to the upper core support plate and the lower core support plate rotation ("rigid rotation").
- the equation used in the program represents the relationship between the stress and displacement, which is deduced from the simple cubic equation of beam.

The equation of lateral deflection w(x) of a beam can be expressed :

$$w(x) = a_0 + a_1 x + a_2 x^2 + a_3 x^3$$

The coefficients a_k are determined from the boundary conditions. Since the stress of a beam is calculated as[6] :

$$\sigma = -ERw''$$

We can get the maximum stress as

$$\sigma_{max} = ER\ \mathrm{Max}(w''(0),\ w''(\ell))$$

,where E=modulus of elasticity,

R=radius of beam, ℓ =length of beam,

w'' =second derivative of deflection w with respect to x.

As mentioned above the program MAIN needs the results of a dynamic analysis using a core model.

The program MAIN needs the following inputs :

- Duration of excitation, total number of nodal points and FA in core model, and total number of spacer grid per FA, nodes of each FA.
- Segment length of FA in core model, outer diameter of guide tubes, and Young's modulus of guide tube material.

The results of program MAIN are :

- The time when the peak stress value occurs.
- The peak stress value at a segment.
- The number of FA and segment at which the peak stress occurs.

The schematic flow diagram of program MAIN is shown in Fig.2.

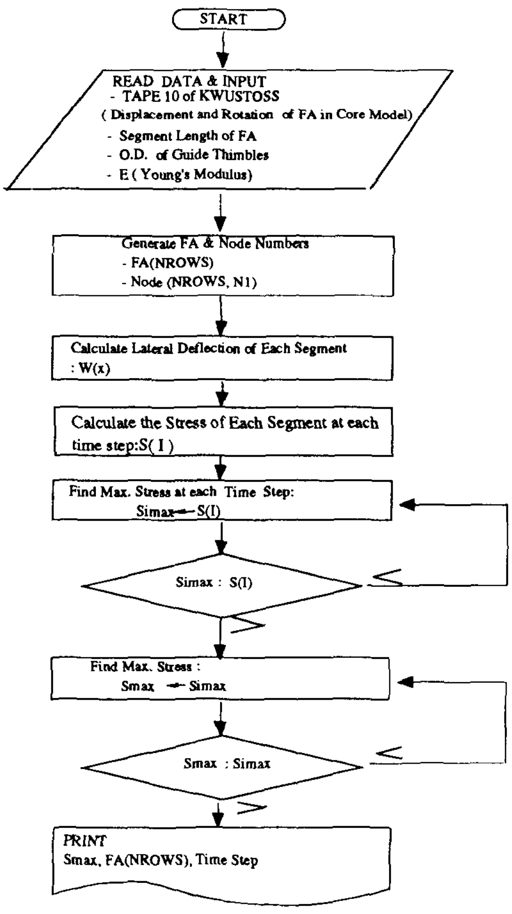

Fig. 2. Schematic Flow Diagram of Program MAIN

2.2. Development of the FA Stress Model

The FA analytical model is developed using the finite element technique, which is done by means of the finite element code KWUSTOSS-STATIC. This model is developed in order to perform detailed stress analysis for the FA components under seismic and blowdown events.

For the development of the model, the model is deduced in 2 steps :

- Skeleton model (FA without FRs).
- Complete FA model.

2.2.1. Skeleton Model

The skeleton model is established using three dimensional beam elements as shown in Fig.3. Each guide thimble in a FA is represented by a beam element with distributed nodes. And the node numbers of each nodal plane, the value of y-coordinate(height coordinate) of the nodes and the numbering of the spacer grid plane are shown in the figure.

The spacers connecting the guide thimbles are modelled as rigid structures by using constraint techniques. Fixed end conditions are used for the skeleton model, which were justified from the FA deflection test (refer Fig.7).

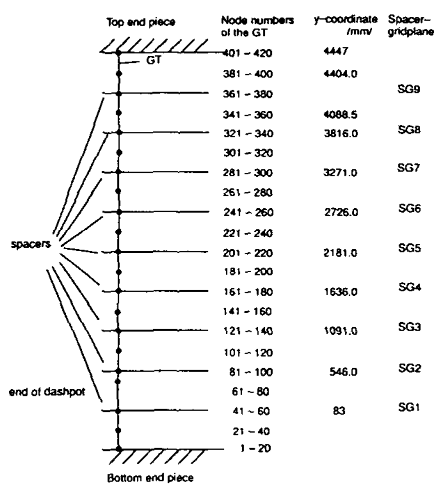

Fig. 3. Skeleton Model

2.2.2. Fuel Assembly Model

In addition to skeleton model the fuel rods are represented by a single row of beams(refer Fig.4). Those beams are connected to one node in every spacer grid plane via linear and rotational springs in order to take account the clamping conditions of fuel rods.

The input data for modelling of the FA stress model are :

- The values of y-coordinate of the nodes determined by the spacer grid position.
- The distributed positions of the guide thimbles in the xz-plane determined by the cross-section of the FA(refer Fig.13).
- The all fuel rods in a FA represented by a single beam.
- The linear spacer grid spring characteristic as shown in Fig.5 taken from the measured data.

  C_{lin} = Spring Force / Deflection
- The rotational spacer grid spring constant(refer Fig.6).

  $C_{rot} = C_{lin} \times \ell^2/4$

  .where ℓ : the distance between the two dimples.
- The material constants of guide thimble and fuel rod, which are the Poisson number and the modulus of elasticity of Zircaloy.
- The results of program MAIN, which are the displacements of the worst bending shaped FA in core model and will be imposed on to the FA stress model as the kinematic constraints.

The results of the FA stress model analysis are :

- The stress at each segment of a fuel rod and guide thimble.
- The displacements and rotations of the spacer grid.

2.2.3. Verification of the Model

For the verification of the established FA stress

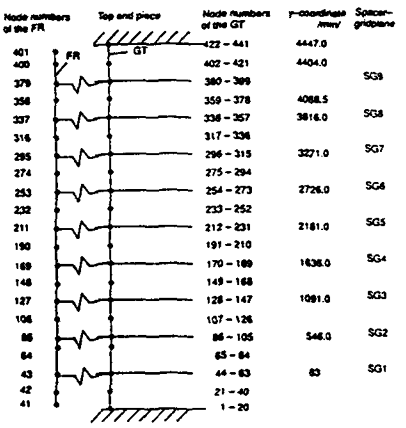

Fig. 4. FA Stress Model

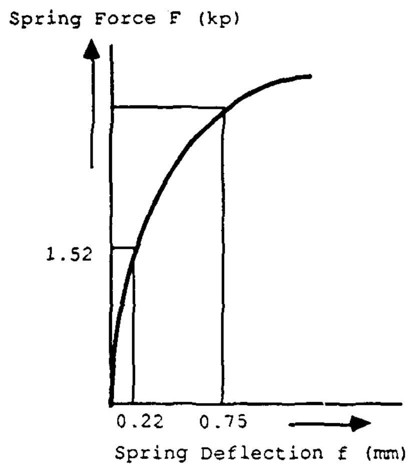

Fig.5. Measured Linear Spring Characteristic(AH76)

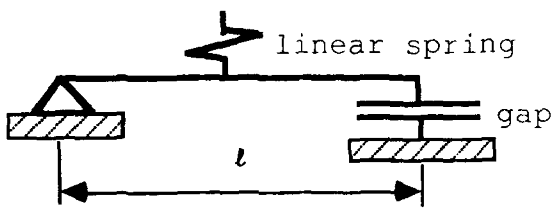

Fig.6. Rotational Spring Model

model. the calculated and the measured lateral stiffness of 16x16 type FA (9 spacer grids) are compared. For this purpose, in case of the skeleton model, a force of 25 N has been applied in x-direction to the four outermost nodes at the 5th spacer grid, which results in a total force of 100N. And in case of FA model a force of 250N in x-direction is imposed on to the four outermost nodes in the plane of the 5th spacer grid, which results in a total force of 1000N. The fuel assembly load-deflection test to get the lateral stiffness of 16x16 type FA is performed . As shown in Fig.7 FA is deflected with hydraulic force at the 5th spacer grid up to 20mm, and at that time the force is measured with the force transducer at the same level.

The resulting deflections at this location are compared with the test results of 16x16 type skeleton and FA as shown in Fig.8 and 9. As it is shown in the figures the test results show non-linear behaviour dependent on the deflection. But since it has been proved experimentally that the behaviour of the FA can be considered to be linear under lower amplitude vibration[7], the calculated result is linearized and averaged for lateral deflection. And as shown in the figures the calculated lateral deflections of both skeleton and FA model are in good agreement with those from the test. Hence both skeleton and FA model can be used effectively for the calculation of the guide thimble stresses for seismic and blowdown events.

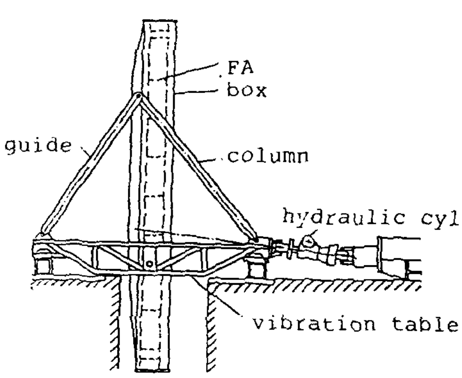

Fig. 7. FA Characteristic Test Set-up.

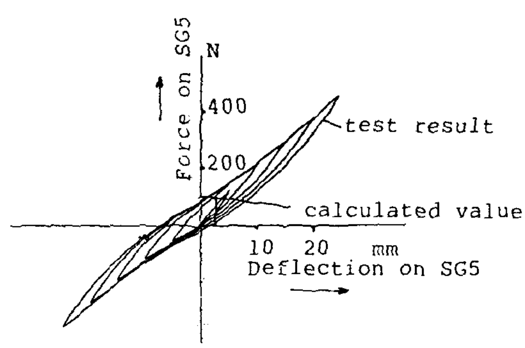

Fig. 8. Lateral Skeleton Stiffness(Test and Calculation).

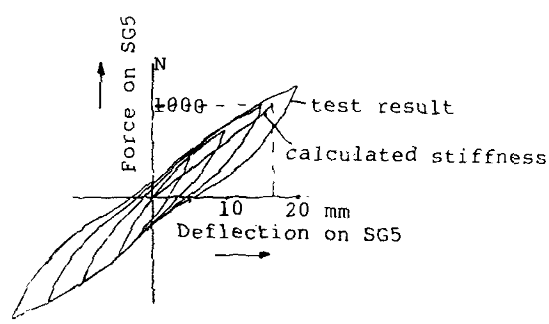

Fig. 9. Lateral FA Stiffness(Test and Calculation)

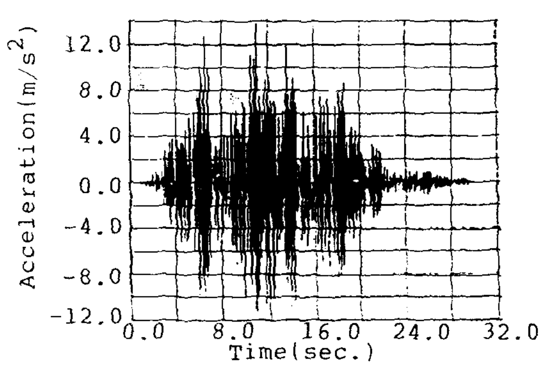

Fig. 10. Arbitrary Seismic Input Time History of Core Plate

3. Stress Analysis of the 16x16 Type FA

As mentioned in previous chapter the stress analysis of the 16x16 type FA under the arbitrary earthquake load, which is shown in Fig.10, is performed using the developed program and model.

3.1. Identification of the Worst Bending Mode FA

The program MAIN uses as input the results of core model analysis which stores on TAPE 10 of KWUSTOSS code. Hence the 9 FA series model as shown in Fig.1 is used as an example. The input of program MAIN to identify the worst bending mode shaped FA in 9 FA series core model is provided as follows :

- Type of problem : lateral, damped
- Initial time of excitation : 0 sec
  End time of excitation : 0.495 sec
  Step size : 0.5×10^{-3} sec
- Nodes fixed with respect to rotation : 3 to 7 and 107 to 151
- Massless nodes : 107 to 151
- Excitation force applied nodes : 1 to 7
- EI for all beam elements : 24500 Nm^2
- *Young's modulus(E) of beam :* 7.98×10^{10} N/m^2
- Diameter of guide thimbles : 0.0138m

As a result of the given input described above the worst bending mode shaped FA is identified as below.

- The time when peak stress occurs : TIMAX = 0.152 sec
- The peak stress at a segment : SPMAX = 6.8528E7 N/m^2
- *The number of FA at which the peak stress* occurs : NUMBER FA = 6
- The number of segment at which the peak stress occurs :
  SEGMENT NUMBER = 4

The displacements and rotations of the worst bending mode shaped FA are also obtained in Table 1. And these values are used as kinematic constraints of the detailed FA stress model.

Table 1. The Displacements and Rotations of the Worst Bending FA

Node	Displacements(m)	Rotation(rad)
LCSP 1	0.37322602E-02	0.20750132E-03
UCSP 2	0.45638907E-02	0.18597255E-03
FA 6 : 63	0.41797476E-02	
64	0.58437975E-02	
65	0.81682980E-02	
66	0.10513935E-02	
67	0.11885101E-01	
68	0.11861356E-01	
69	0.10946024E-01	
70	0.92484022E-02	
71	0.73286112E-02	
72	0.58096654E-02	
73	0.48285035E-02	

3.2. Stress Analysis of the 16x16 Type FA

The objective FA(16x16 type FA) is shown in Fig.11. And the stress model according to the modelling method described in chapter 2. in which the guide thimble and fuel rods are represented to the 3-dimensional beams and those beams are connected in every spacer grid plane via linear and rotational springs, is shown in Fig.12. From the Fig.11 the position of node in y-coordinate can be determined. The distribution of the guide thimbles in xz-plane is shown in *Fig.13. The representative fuel beam is placed in* the middle of the model(x=z= 0.0).

The input of FA stress model is as follows :

- Cross-sectional area and moment of inertia of fuel rods :
  A=205x22.83=4680.15 mm^2

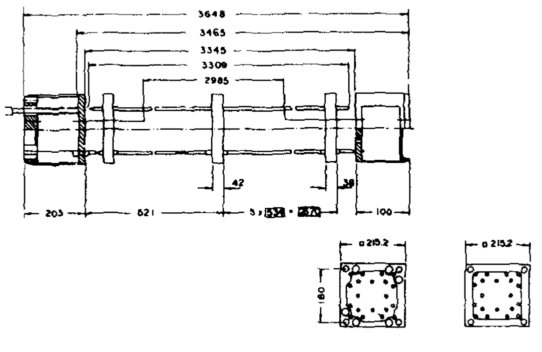

Fig. 11. 16 x16 Type FA

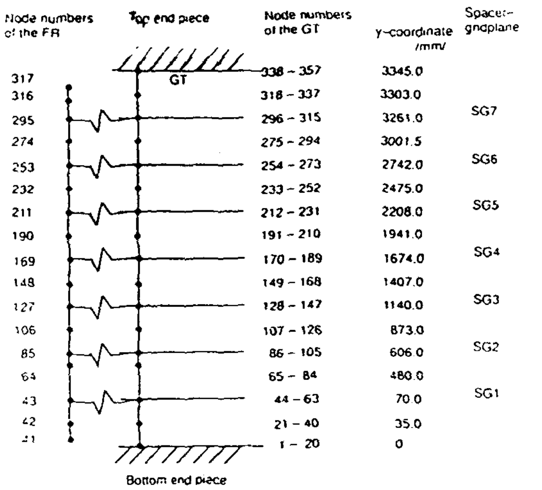

Fig. 12. 16x16 Type FA Stress Model

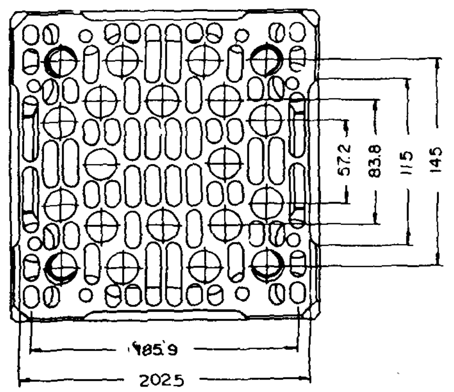

Fig. 13. Distribution of the Guide Thimble in the XZ-Plane

$I = 205 * x288.35 \approx 59111.75\ mm^4$

* The numbers of fuel rods of the FA = 205 fuel rods

- Linear spacer grid spring characteristics at operating temperature : $C_{lin} = 205x0.92x35/0.825\ N/mm = 8000.4\ N/mm$
- Rotational spring characteristic : $C_{rot} = 1.80x10^6$ N-mm/rad
- The material properties for Zircaloy of the guide thimbles and the fuel rods : Modulus of elasticity(E) at operating temp. $= 0.798x10^5\ N/mm^2$
  Poisson ratio $\nu = 0.3$
- The displacements and rotations obtained by the program MAIN are imposed on to the FA model as the kinematic constraints.

4. Results and Discussion

Based on the worst bending mode shaped FA in core model the 16x16 type FA stress distribution under arbitrary seismic load is obtained. In the detailed stress model the rotations of the spacer grids are calculated more realistically corresponding to the displacements of the spacer grids, which results in the value of $50.93\ N/mm^2$ in the 3rd spacer grid plane, than those in the reduced FA model ,which is calculated to $68.5N/mm^2$ between 2nd and 3rd spacer grid. And this stress value obtained from the detailed stress model will be lower than the acceptable limits of $125N/mm^2$ which is the material yield strength of the guide thimble, thus the guide thimble will not buckle elastically.

The program MAIN and the detailed stress model to calculate the stresses in FA components for seismic and blowdown events are developed. Hereby throughout the developments the following results can be deduced :

- The developed FA stress model is very helpful for the stress calculation of a FA components for the postulated seismic and blowdown events.

–The stress analysis of the FA for seismic and blowdown events are performed by checking of the guide thimble stress whether the FA satifies the ANSI design condition III and IV or not.

–The assumptions, kinematic constraints and the equation for modelling of FA are justified.

References

1. "American National Standard for Light Water Reactors Fuel Assembly mechanical design and evaluation", ANSI/ANS-57.5-1981, American Nuclear Society, 1981.5
2. "Preliminary Safety Analysis Report for Korean Units 7&8", Korea Electric Company.
3. "Preliminary Safety Analysis Report for Yonggwang 3&4", Korea Electric Power Corporation.
4. "Reload Transition Safety Report for YGN 1&2", SIEMENS/KAERI, 1988.
5. Stabel and I.K.Kim, "KWUSTOSS-STATIC Code, "How to use" Part", Siemens/KWU, 1991.
6. S.P.Timoshenko and J.M.Gere, "Mechanics of Materials", Litton Educational Publishing. Inc., 1972.
7. A.Preumont, "A Two-Time Step Algorithm for Seismic Analysis of a PWR Core", Nuclear Engineering and Design 65,1981, P.49-62