

A 3-D MODEL WITH SUPERELEMENTS FOR DYNAMIC ANALYSIS
OF A PWR VESSEL INTERNALS

Carlos Alexandre de J. Miranda
IPEN - Inst. de Pesquisas Energ. e Nucleares
Travessa R, 400 - CEP 11049
Cid. Universitaria - Sao Paulo - SP (BRAZIL)

ABSTRACT

In order to obtain an integrated 3-D model to allow its dynamic analysis, all the internal structures of a PWR vessel were modelled by superelements. In the GENERATION PASS the superelements were defined, basically, with the STIF63, STIF4, STIF21, STIF27 and STIF45 elements of the ANSYS 4.4A library. In this pass, the IN-PLANE ROTATIONS were the only degrees of freedom (DoF) restricted for the plane superelements and no loads were applied. The stiffness and the mass matrices were saved and condensed for the defined master DoF (the consistent mass matrix concept was used for the STIF63 elements). In the USE PASS, the superelements were put together with other "single" elements, the boundary conditions were applied and the loads were defined. The model will be filled with more than 300 gap elements if the non-linearities become present. There are 90 superelements in the 3-D model having, in total, about 85000 DoF and 8200 masters DoF with about 12000 elements (about 8400 shell elements). The model was developed and run on a VAX 11/785 computer. Some superelements needed about 2 hours of CPU time to be generated.

The behavior of the complete 3-D model was tested with ACEL static load (KAN,0), without gaps, with 3 load-steps and used about 3:00h of CPU time to run. In a dynamic analysis (KAN, 2 with 3 spectra), it has used about 6:00h of CPU to run. The results have shown good agreement with the expected behavior for the structure under these loads. For the seismic loads the model remained linear. For other dynamic loads, higher than the seismic ones, the model will be filled with gaps and a non-linear analysis (KAN,4) performed.

INTRODUCTION

The Internals structures of a PWR vessel were analysed first, isolated from the others, with static ACEL loads [1], in the horizontal and vertical directions. In the Internals there are over 300 gaps, principally among Fuel Elements (FE) and between FE and the Shrouds. So amplifications may occur due to the impacts among gaps and due to the dynamic characteristics of the loads. With a high excitation level the behavior of the Internals will be well determined only with a non-linear analysis. For that analysis a 3-D model was developed considering all the structures together as superelements and the existing gaps as gap elements, with the aid of the ANSYS 4.4A program [2]. Currently the first scope of this work is the analysis under the seismic loads defined by the acceleration response spectra given in the figure 3, with the other dynamic loads considered later.

The gaps in the developed 3-D model were deleted in the test analyses with static ACEL loads, in the frequency test analyses with masters degrees of freedom (DoF) defined only in Z (or only in X) direction and in the seismic spectral analysis with DoF masters in X, Y

relative displacements obtained from the nodes in the gap positions. If they are greater than the gap values the analysis must be changed to a non-linear one (KAN,4) with the gap elements activated and the application of the displacement pulse derived from the accelerograms obtained from the seismic acceleration response spectra. If they are lower than the gap values the spectral analysis will continue to obtain the displacements and stresses inside the superelements (STRESS PASS) for each significant node and, then, combined properly.

GENERAL DESCRIPTION. The modelling of the Internals was conceived with the desire to avoid simplifications which should need a long and expensive experimental program to validate the assumed hypotheses. Other than its complexity and size the developed model has, in evidence, the extensive use of 3-D gap elements (STIF52) and superelements (STIF50). The superelement has associated one stiffness matrix and one mass matrix and there is no geometry once formed. It can be connected with other superelements and/or 'single' elements by its master DoF. Its matrices remain unchanged so it is linear. Any non-linear element used to create it remains with its initial STATUS used in the creation run (GENERATION PASS). In the present model the superelements were formed only by linear elements. In the GENERATION PASS any load type can be applied but in the already created superelement (USE PASS) only forces and displacements can be applied in the master DoF (other than accelerations). The Reactor Pressure Vessel (RPV) will be considered rigid to the planned analyses and modelling scope.

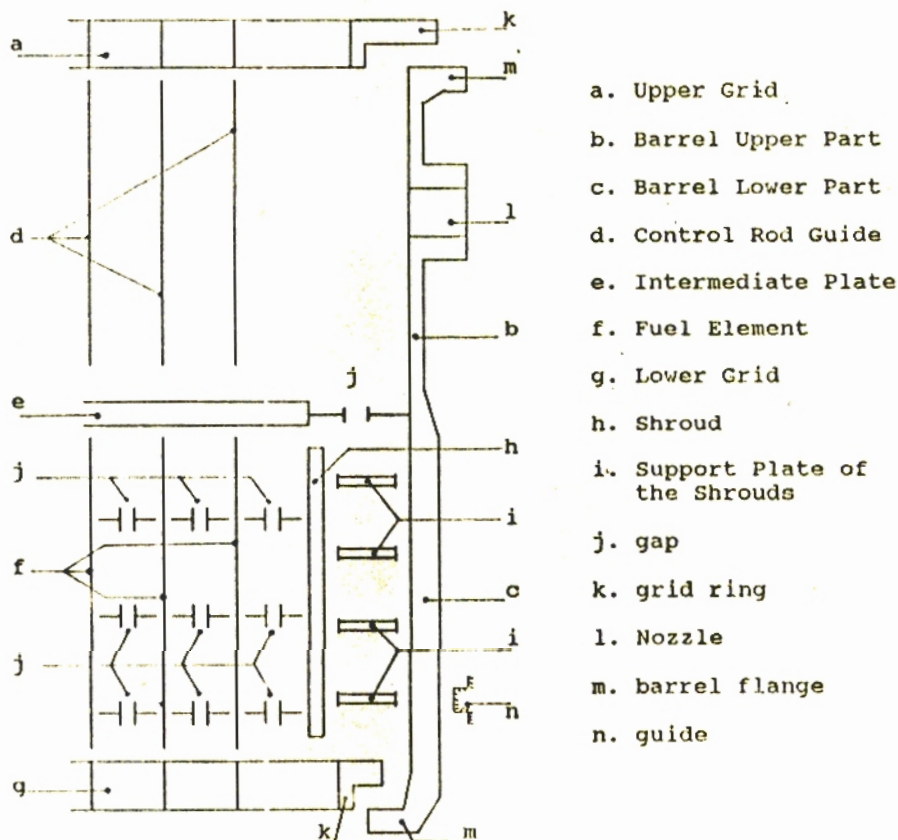


Figure 1: Relative positions of the Internal Structures

OVERALL BOUNDARY CONDITIONS. In the figure 1 is presented the set of the modelled structures forming the internals and their relative positions. All the nodes representing the RPV, in the developed 3-D model, were clamped. In the following the existing connections with the RPV will be defined with the adopted boundary conditions: .1. In the Upper Part of the Internals: a) there are special connections between RPV-Core Barrel-Upper Grid to restrain the tangential relative displacements. These connections were modelled with fictitious rigid truss elements with no mass, tangents to the Core Barrel (BARREL) and to the Upper Grid and b) the upper flange of the BARREL is supported in the flange of the RPV so the vertical displacements were restrained in it; .2. In the Nozzles: There are gaps modelled with gap elements between the nozzles of the BARREL and RPV; .3. Lower Guides. There are radial gaps between the BARREL and the RPV modelled with gap elements.

GENERATION PASS

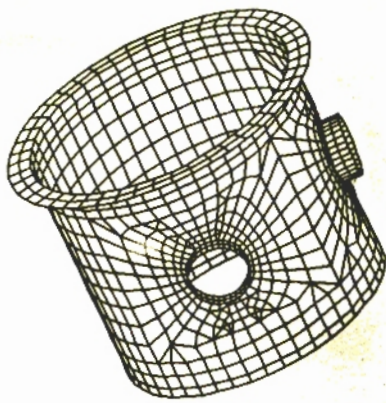
Basically the following elements were utilized: STIF4 (3-D beam) - to discretize the Control Rod Guides (CRG) and used also in other superelements as auxiliary in the modelling; STIF8 (3-D truss) - as auxiliary to the modelling; STIF21 (concentrated mass) - with KEYOPT,3,2: 3-D mass without rotary inertia; STIF27 - to introduce in the model the stiffness of the Fuel Element (FE) calculated in (3). This element does not have geometry and is defined by 2 nodes with 6 DoF each. Only the upper half of the matrix is given by "REAL CONSTANTS"; STIF45 (3-D Solid 8-node) - used in the Upper and in the Lower Grid Rings; and STIF63 (4-node general shell) - to model the major part of the structures as the BARREL, the Intermediate Plate (I.P.), the Shrouds and the Support Plates of the Shrouds (SPS) and part of the Upper and the Lower Grids.

SUPERELEMENTS DESCRIPTION. Table 1 presents some parameters of each superelement. The only boundary conditions defined in this phase are: (a) the "IN-PLANE ROTATIONS" for the plane superelements and (b) the ROTZ rotations for the Fuel Elements. No loads were defined. Some of the modelled superelements are presented in figure 2.

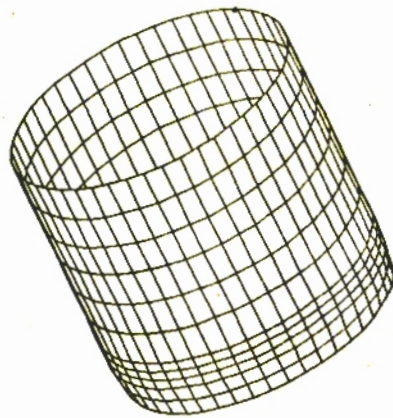
SUPER-ELEMENT	Number of		Max. Wave Front	Active DoF	Mass (Kg)	Number of Super.	CPU TIME (s)
	Node	Elem.					
BARILSUO	1207	580	534	436	1445	01	2800
BARILSU1	1207	580	462	436	1445	01	2400
BARRILIF	672	624	980	840	6298	01	6400
UPP GRID	860	652	522	348	1295	01	1800
LOW GRID	839	570	312	141	718	01	780
C ROD G	150	132	168	18	50.3	21	270
PLACAORI	1451	662	320	257	73.5	01	2100
PLACAREF	1451	662	320	257	73.5	01	1160
FUEL EL	10	29	38	28	223.5	21	17.4
SUP PL 1	168	130	102	81	8.9	04X4	59.0
SUP PL 2	108	74	137	83	5.1	04X4	41.0
SHROUD 1	121	100	234	199	124.9	04	194
SHROUD 2	110	80	311	280	72.8	04	213

TABLE 1: Parameters of the Superelements (Generation Pass)

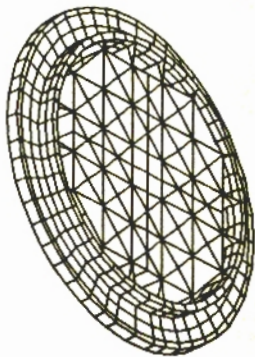
The following superelements were created: (a) BARREL - the Core Barrel was divided in two parts: the UPPER PART (divided in two symmetric sectors: between 0° and 180° (named BARILSUO) and between 180° and 360° (BARILSU1) and the LOWER PART (BARRILIF), (b) UPPER GRID - The Grid was modelled with STIF63 shell elements and for the ring was used



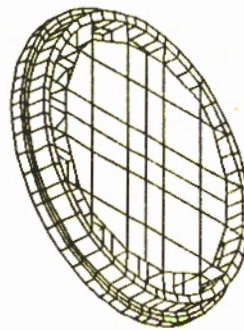
(a) Barrel Upper Parts



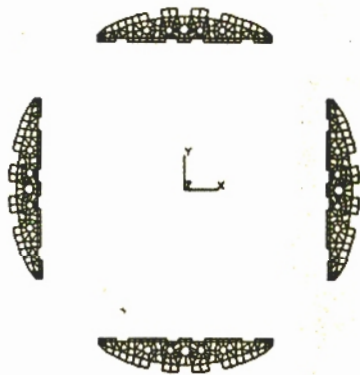
(b) Barrel Lower Part



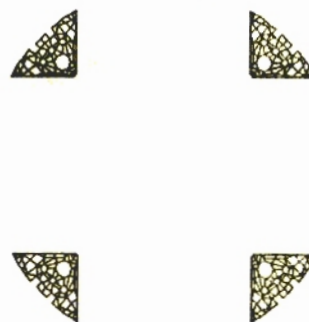
(c) Upper Grid



(d) Lower Grid



(e) SPS Type 1



(f) SPS Type 2

Figure 2: Some of the Modelled Superelements

the STIF45 element. Some rigid 3-D beam elements without mass were used, also, to make compatible the rotations between elements with different DoF per node. The perforated plates were modelled without holes using "equivalent" values for density, YOUNG's Modulus and POISSON's Coefficient; (c) CONTROL ROD GUIDE (CRG) - formed from 20 simple-guides and 4 double-guides. There were some perforated plates along the structure modelled as rigid region; (d) INTERMEDIATE PLATE - for computational and modelling convenience this plate was modelled in two identical but symmetrical parts: between 0° and 180° (PLACAORI) and between 180° and 360° (PLACAREF); (e) LOWER GRID - this grid, with its ring, was modelled like the upper grid. There is a flow uniformization skirt, with no structural function, modelled as concentrated mass and some octogonal shaped plates, modelled as 3-D beam elements, where the FE pins are fixed; (f) FUEL ELEMENT (FE) - the nozzles were modelled with 3-D beam elements and each part between nozzles has its stiffness matrix given explicitly by the STIF27 element; (g) SHROUDS - the shrouds, positioned around the Core, form a guide for the water flow. There are two basic groups: the SHROUDS Type 1 (4 plates) and the SHROUDS Type 2 (8 plates coupled 2 by 2 in a 90° angle) associated with the respective Support Plates. Four Superlements SHROUD Type 1 and 4 SHROUD Type 2 (each one grouping 2 Shrouds Type 2) were created; (h) SUPPORT PLATES of the Shrouds (SPS) - they are positioned in 4 levels along the Reactor Core length. In each level the SPS are positioned around the Core and, in order to allow this, there are 2 basic types or groups: the SPS Type 1 (positioned at 0°, 90°, 180° and 270°) and the SPS Type 2 (at 45°, 135°, 225° and 315°). All these superlements were defined not to work with a rotated set of superlements with respect to the original defined ones. The same applies to the Shrouds.

USE PASS - 3-D MODEL DESCRIPTION

In the "USE PASS" the following elements were used, from ANSYS 4.4A library, other than the STIF50 superlements: STIF4 - to model the pins between the I.P. and the BARREL; STIF8 - to model the springs between the I.P. and the FE with properties and Initial Strain defined in order to reproduce the pre-compressing forces (Fcomp=3000 N) created during the overall assemblage. They are used also to simulate the connection RPV-Upper Grid-BARREL in the top part of the model; STIF52 (3-D Gap) - used to model the gaps existing among the Internal structures.

CONNECTIONS AMONG SUPERELEMENTS - Some of these connections may be seen in figure 1. One supposes that there is continuity between the Upper Grid and the CRG. The same applies to the CRG-I.P. connection. The Lower Grid is connected to the lower flange of the BARREL by pins and bolts, so continuity in displacements only is supposed. The same applies to the connections between the SPS and the respective SHROUDS and those between the BARREL and the SPS. There are 6 spacing grids for each FE with nominal gaps of 0.5mm between FE and SHROUDS and nominal gaps of 1.0mm among FEs. Between the Nozzles of the BARREL and those of the RPV and between the RPV and the BARREL, in the level of the guides, in the radial direction, there are theoretical gaps of some tenths of millimetre.

The other connections are: (a) BARREL-RPV-Upper Grid. Among the ring of the Grid, the flange of the BARREL and the RPV there are 4 connection points: in these points the radial displacements are free and the tangential ones are restrained; (b) I.P. and BARREL - In this connection there are: .1. four pins equally spaced, connected to the BARREL and inserted in the I.P. (with a theoretical null gap) and .2. a nominal gap of 0.5mm between the I.P. and the BARREL only in those positions associated with the pins; (c) I.P. and FE - for these connections there are pins and springs which react against the I.P.. And, after the overall assembly, there is a gap of 1.0mm limiting the spring deformation; (d) FE and Lower Grid - The lower nozzle of the FE reacts against the Lower Grid with a null gap and there are pins to couple the transverse displacements; (e) SPS Type 1 and Type 2 - In each

level the SPS Type 1 border is at the side of the SPS Type 2 border with a nominal null gap. These gaps are not modelled.

USE PASS - TEST ANALYSES

Some tests had been done to control and assure the good behavior of the developed 3-D model with the described Superlements and single elements.

STATIC ANALYSES - Several parameters were controlled and, in particular, the control of the "RANGE OF ELEMENT STIFFNESS" and "EQUATION SOLVER PIVOT" values showed the possibility of numerical errors in the results due to the low values of the stiffness associated with the "IN-PLANE ROTATIONS" of the plane superlements. Because of that these rotations were restrained in the re-definition of the superlements and the problem was minimized. Another problem identified in this test phase was the non-equilibrium of the superlement FE stiffness matrix, initially defined with 4 significant figures. This problem was solved by the use of 6 significant figures. Once these "macro-problems" were resolved the analysis of the results showed good agreement with what was expected taking into account the nature of the structures and the applied loads (ACEL loads only).

DYNAMIC ISOLATED ANALYSES - Initially the frequencies of the following isolated structures were calculated, without considering the influence of the fluid (added mass): FE, CRG, SPS, SHROUDS types 1 and 2. The applied boundary conditions simulate the INTERNALS connections, including the mass contribution of the structures which are supported by others. The BARREL was analysed also but considering only its upper support with 2 axisymmetric harmonic models: (1) with 2-node shell elements considering "added masses" and (2) with 4-node solid elements modelling also the fluid. These analyses are described in (4). For the CRG structure 2 hypotheses were considered for its lower end: a) only the vertical displacement is free and b) the displacements are free and the rotations restrained. The other structures are considered to be much more rigid than these ones. The results are presented in table 2.

Super-Element	mode / frequencies (Hz)					Notes
	1	2	3	4	5	
BARREL	35.1	55.6	100.	124.	139.	model 1
	32.7	69.9	126.	154.	180.	model 2
C.R.G.	75.2	75.2	---	---	---	cond. a
	186.	186.	---	---	---	cond. b
SHROUDS	210.	230.	312.	331.	399.	TYPE 1
	209.	229.	402.	546.	---	TYPE 2
S.P.S.	81.	298.	329.	503.	586.	TYPE 1
	29.8	490.	826.	836.	1321	TYPE 2
F.E.	5.84	27.2	27.2	50.2	50.2	-----

TABLE 2: First Frequencies of the Isolated Internal Structures

The first frequency of the SPS Type 2 is a transverse mode, lower than 33 Hz but near this limit. Therefore it can be considered rigid to the seismic excitation because the used model to calculate its frequencies is more flexible than the reality (the same applies to the SPS Type 1). For the FE the frequency of 5.84 Hz is associated with its spring in the top. Since the first real bending mode is under 33 Hz, a simplified seismic analysis will be made in it as follows.

FUEL ELEMENT. Isolated Response Spectra Analysis - Since the FE has presented frequencies below 33 Hz the same model was used to make simplified analysis with the acceleration response spectra, given in figure 3, applied in two load steps: for X and for Z directions.

. X excitation: The first 2 transverse modes (modes 2 and 3) have associated respectively 36% and 43% of the FE mass. Considering the applied excitation the maximum modal displacements are 0.0236mm and 0.0280mm respectively. These are the more significant modes. Table 3 presents the results of the SRSS combination for the 5 significant modes using FACT=0.001 in the ANSYS EXMODE command.

. Z Excitation: As was expected in this case the first mode (5.84 Hz) has all the FE mass associated with it. The maximum Z displacement is 4.14mm (as it will be presented in the 3-D model seismic analysis, with the gaps deleted). In this analysis/model the FE in Z direction is connected only in its upper part by its spring) and the maximum reaction force is 1240 N. The spectral acceleration associated is 5.5m/s².

mode	Hz	acel	R _{x,i}	R ² _{x,i}	R _{y,i}	R ² _{y,i}
2	27.2	1.21	90.8	8253	-99.1	9823
3	27.2	1.21	108.1	11694	99.1	9823
7	82.5	1.17	-17.8	318	2.8	8
11	122.9	1.17	-8.3	70	-0.2	0
13	186.8	1.17	23.3	544	0.3	0.1
SQRT(sum of R ²)				144.5 N		140.2 N

Table 3: FE SRSS combination (X acceler. spectrum)

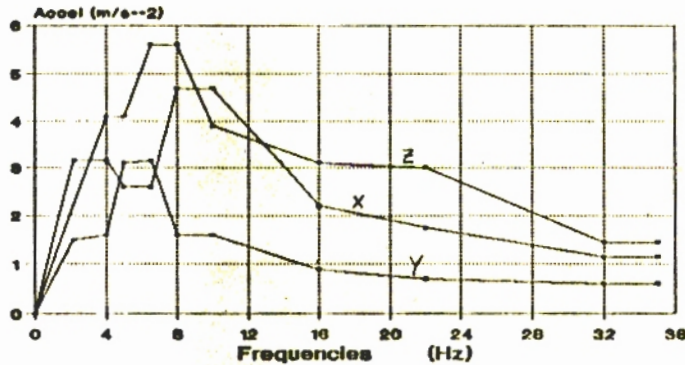


Figure 3: Acceleration Response Spectra

DYNAMIC ANALYSES. 3-D MODEL. For the entire 3-D model the overall frequencies were calculated in two test analyses defining the master DoF first only in Z direction and, after, in X direction, with no masters in the I.P., SHROUDS and SPS. The Householder procedure was used to extract the modes associated with the 120 masters DoF (defined in each analysis at the same nodes). Some of the results are shown in the table 4.

The results show good agreement with the expected ones. Some of the frequencies are repeated or are close to those in table 2 and some of the calculated modes present a multiplicity of 21, again as expected, like: (a) the FE spring modes (5.84 Hz); (b) the transverse FE modes (21

modes ranging from 26Hz up to 33 Hz and 21 others ranging around 74Hz, respectively the 1st and the 2nd bending modes of one 'single' FE); (c) the FE modes in Z direction and (d) the 1st flexional mode of the CRG (ranging from 200Hz up to 330Hz).

mode	frequency	remarks	mode	frequency	remarks
1 to 21	5.84	FE springs	1	26	FE
25: 26	104: 123	UPPER GRID	2 to 21	32.6	FE
27	132	LOWER GRID	22	40.6	FE
28: 30	211: 275	UPPER GRID	24 to 44	74	FE
31: 33	288: 319	UPPER GRID	46	151	BARREL
35	332	UPPER GRID	47: 48	202	CRG
37: 38	361: 381	UPPER GRID	50: 52	256	CRG
61 to 81	846	FE	53:	300	CRG
			55 to 60	300	CRG
Masters DoF only in Z			Masters DoF only in X		

Table 4: Some frequencies (Hz) of the overall 3-D Model (Test Analyses)

So, excluding the FE, and without the added mass effect, all the Internals can be considered rigid for the seismic excitation. Therefore a dynamic seismic (spectral) analysis was made to calculate the behavior of the structures and to verify if it remains linear under this load.

SEISMIC ANALYSIS. 3-D MODEL

Once assumed the good overall behavior of the developed 3-D model the spectra was applied, in 3 load steps (one for each spectrum direction), with the significance factor of 0.002 to expand the significant modes for each direction, with no gaps. The defined 360 master DoF are on the same 120 nodes chosen during the tests (but with the X, Y and Z displacements as masters). The Subspace Iteration procedure was utilized to extract the 120 first modes. After this the POST27 was run to combine properly the modal results of each load step and among load steps.

The SRSS - Square Root of the Sum of the Squares - method was chosen to combine the modes and the load steps. This procedure, before the STRESS PASS applied over each expanded mode, was intended to verify if the maximum X and Y displacements, in the gap positions, are lower or greater than the existing gaps. The defined masters capture about 90% of the total mass in the 3-D model.

Some of the frequencies are presented in the table 5. The more significant modes in each direction are presented in the table 6 with the associated mass. The maximum displacements in the X, Y and Z directions are presented in the table 7.

The result analysis showed that the maximum displacements are lower than half of the gap values, so the gaps remain open and the structure remains linear during the seismic excitation. From the tables 6 and 7 the results confirm the conclusion made in the previous discussions, i.e.: the 3-D model, the structure, is rigid to the seismic excitation. Further discussion is made in the next item.

modes	1 to 21	22:23	24 to 63	64:65	66 to 105	110
frequency	5.82	26.1	32.6	40.8	74.0	104.

Table 5: Some Frequencies of the 3-D Model (Hz)

mode	mass fraction	mode	mass fraction	mode	mass fraction
22	59.2%	23	57.2%	1	28.7%
65	25.9%	66	24.5%	110	32.9%
106	3.5%	107	3.5%	112	38.3%
X excitation		Y excitation		Z excitation	

Table 6: Some significant modes in each direction with the associated mass fraction

excitation	X	Y	Z
X	0.084*	0.002	0.010
Y	0.001	0.045*	0.004
Z	0.005	0.005	4.36*
SRSS	0.084*	0.045*	4.36*

Table 7: Maximum Seismic Displacements (mm)
(*) - in the Fuel (Super) Element

FINAL REMARKS

The tests showed that the developed 3-D model has a good behavior, compatible with the expected one for the applied loads. The calculated frequency values are those, or near those, expected values with some modes presenting a multiplicity of 21. The transverse modes are practically repeated in the X and Y direction because, as can be seen in the figures, the model is almost symmetric with relation to the two orthogonal vertical planes. Thus, the model, without gaps in a first approach, is good for the static and dynamic analyses (seismic) to be made in the design phase of the Internals.

The Internals are rigid for the seismic excitation and the frequency below 33 Hz, associated with the EC is due to the particular vertical coupling condition assumed for it (with free vertical UZ displacement in its lower part). But it can be shown, from the results, that the earthquake excitation is insufficient to separate the FE from the Lower Grid because the maximum force in its spring (F=1240 N), due to the earthquake, is lower than the pre-compressing forces of the spring (3000'N).

From these facts the design for the seismic loads can be made with equivalent static analyses. In other words, the first analyses (1) made with the ACEL loads, greater than the maximums given in the figure 3, are enough to demonstrate the good design of the internals structures under the seismic loads.

With respect to other dynamic loads, if they are more severe than the seismic ones, it will be necessary to work with a second level of superelements in a non-linear analysis with the model filled with the gap elements to consider the impacts among the Internals.

REFERENCES

1. IPEN Internal Report R522.03.014.442.10/00. "Conclusao do Dimensionamento Estático dos Internos". Sao Paulo, JAN/1990
2. ANSYS USER'S MANUAL - Version 4.4A
3. IPEN Internal Report RP22.03.0141.721.01/00. "Análise ao Choque dos Elementos Combustíveis". Sao Paulo, Dec/1990
4. IPEN Internal Report R522.03.014.442.02/00. "Cálculo das Frequências Próprias do Barril Considerando o Efeito do Fluido". Sao Paulo, Set/1988