2009 International Nuclear Atlantic Conference - INAC 2009
Rio de Janeiro,RJ, Brazil, September27 to October 2, 2009
ASSOCIAÇÃO BRASILEIRA DE ENERGIA NUCLEAR - ABEN
ISBN: **978-85-99141-03-8**

TRANSIENT THERMAL ANALYSIS OF A 1:2 SCALE CASK FOR RESEARCH REACTORS NUCLEAR SPENT FUEL ELEMENTS CONSIDERING THERMAL CONTACTS AND IRRADIATION

Carlos A. J. Miranda[1], Gerson Fainer[2], Miguel Mattar Neto[3]

[1, 2,3] Nuclear and Energy Research Institute, IPEN-CNEN/SP
Av. Lineu Prestes, 2242 (São Paulo, SP, Brazil)
[1]cmiranda@ipen.br, [2]gfainer@ipen.br, [3]mmattar@ipen.br

ABSTRACT

This work shows the approach used to the numerical simulation of the thermal test of a 1:2 scale model of a dual purpose cask (transportation and/or storage) for spent fuel elements from nuclear research reactors. Conservatively, the cask impact limiters are not modeled. This test is part of the requirements for the qualification of transportation packages for nuclear reactors spent fuel elements. Also, it is part of an IAEA sponsored project which includes Latin American countries with research reactors. This cask model has a stainless steel double wall cylinder (which contains the biological lead shielding) with flat heads and internal structures to accommodate the fuel elements. The cask project is described briefly as well as the developed finite element model and the main adopted hypothesis to consider the non-linearities as thermal contacts, properties varying with the temperature, phase change (thermal shielding lead) using the enthalpy method, and radiation among the internal parts. The analysis will cover the 30 min heating condition at 800 ºC and about 2 hours of the cooling phase. As the main purpose of the paper is to present the proposed approach for the thermal test numerical simulation, only some preliminary numerical results are shown without any comparison to the experimental ones.

1. INTRODUCTION

The main purpose of the regulations [1] that guide the cask project for radioactive material transportation is to protect the people, property and environmental protection from the effects of the irradiation. It is required the containment of the radioactive contents, the control of external levels of the radiation from the cask inside, the prevention of criticality, and the prevention of damage caused by heat, under normal and accident conditions.

The hypothetical accidental conditions, based on a scenario accident, that transportation casks for spent fuel elements from nuclear reactors should resist were already described in details elsewhere [1, 2]. Among other, the main conditions are: a free drop for nine meters on a rigid surface, an 800 ºC fire for thirty minutes and immersion at 200m during one hour. The cask qualification is experimental and the tests should be performed in sequence using the same (damaged) cask. The use of cask impact limiters is allowed. Besides the assurance against criticality of the fuel elements and biological shielding, the (damaged) cask should maintain its structural integrity assuring no leakage in any condition. The 9m free drop is the structural sizing condition. It is possible to test models in 1:2 until 1:4 scales. Although the experimental qualification is strongly recommended, besides the overall sizing verification usually numerical analyses are preliminarily done to help the experimental work, as a guidance to determine the most critical position for the free drop tests and to predict the temperature values inside the cask in the fire condition. Also, many times a combination between numerical and experimental methods is used for casks qualification and the validation of the numerical simulations may be developed by using scale models tests. This is the case of the present work.

There is an IAEA sponsored project to develop a dual-purpose cask for transportation and storage of spent fuel elements from research reactors which includes some Latin American countries with this type of reactors. In previous works presented elsewhere [2, 3] the 9m free drop test was numerically simulated. The results adhered very well with the experimental ones. Also, a discussion on the mechanical nonlinear issues, as the contacts and material properties were addressed. This work deals with the 1:2 scale cask model thermal test numerical simulations. It describes the developed numerical model, the main adopted hypothesis to consider the non-linearities as thermal contacts, properties varying with the temperature, phase change (lead) using the enthalpy method and radiation among the internal parts. As the main purpose of the paper is to present the proposed approach for the thermal test numerical simulation, only some preliminary numerical results are showed without any comparison to the experimental ones. With the need of some calibration in the model and material properties, as the thermal contact conductance, to predict the results in the prototype field tests some comparisons between the numerical results and the experimental ones for scale model tests will be conducted in near future.

2. THE 1:2 SCALE CASK MODEL DESIGN DESCRIPTION

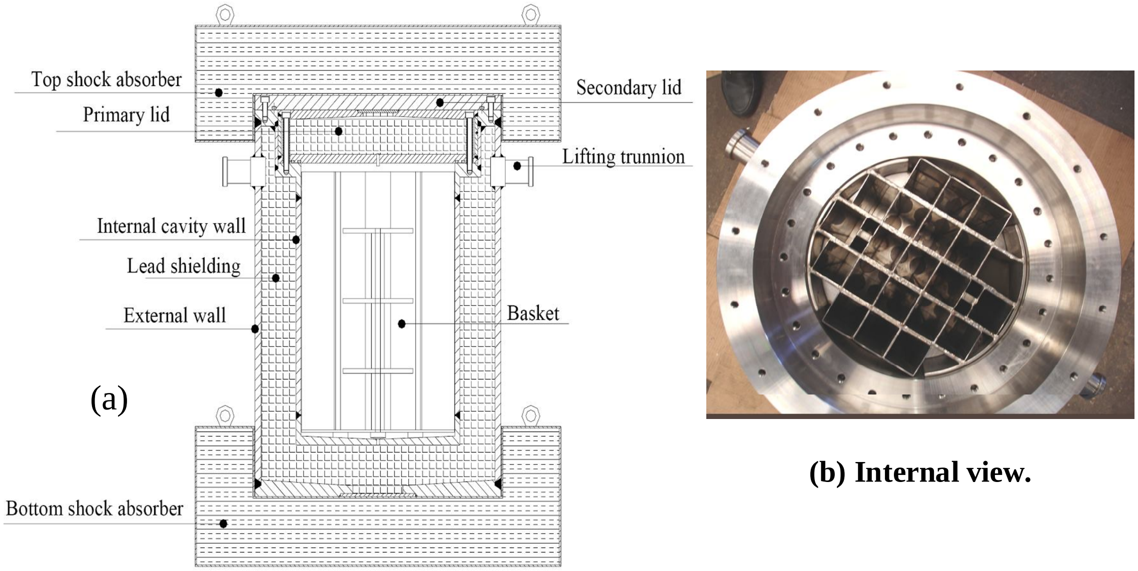
Internally the cask has a basket that can be changed accordingly the spent fuel elements type being transported. It has, also, two concentrically stainless steel cylinders connected by two flanges (internal and external) with the cavity between them filled with lead and a double shell upper closure, also filled with lead, fixed to the internal flange by bolts with a metallic o-ring to assure the tightening. The lead is the biological shield against the radiation. There is, also, a plate connected to the external flange by bolts to fix the upper closure. To reduce the deceleration level in the cask inner parts there are two impact limiters, constituted by Oriented Strand Board (OSB) glued plates and surrounded by a thin stainless steel shell. They are connected to each other by four round bars. Figure 1 shows the main parts of the cask model.

Figures 1.b is a photo of an internal view of the cask half scale model with the basket in its position and the holes for the bolting connection in both flanges. This basket is one designed to accommodate 21 fuel elements from the IEA-R1 research reactor. It is formed from several stainless steel tubes of square section and 2mm thick and four plates among them. All parts are put together by direct welds and by small steel pieces in the corners and positioned in three levels. These peaces are depicted in figure 2 and are shown in details in figure 3. There are, also, two lateral curved steel plates to make the set stiffer. In the bottom there is a circular plate with holes. There are, also, three circular pieces of steel welded in the circular plate to maintain the basket set at 10mm from the bottom of the internal cylinder. There is a gap of about two mm between the basket and the internal cylinder wall.

The cask model has a diameter of about 0,50m (external cylinder) and it is about 0,60m high without impact limiters. With the impact limiters the overall dimensions are about 0,90m (external diameter) and 1,00m high. For the thermal test and so, for the correspondent numerical simulations, these cask impact limiters are not considered (a conservative hypothesis).

3. NUMERICAL MODEL DESCRIPTION

Figure 2 shows the developed finite element model using the solid elements with 20 nodes each – SOLID90, from the ANSYS program [4] library, with diagonalized specific heat matrix with two symmetries in the structure (which is an approximation adopted for fast processing in this preliminary analysis). For thermal radiation the SHELL57 element was adopted with unit thickness, null density and emissivity as defined in table 1 to create the radiation matrix as Superelement and, after that, the shell elements are deleted. For the thermal contacts the TARGE170 and the CONTA174 elements were used with quadrilateral shape.



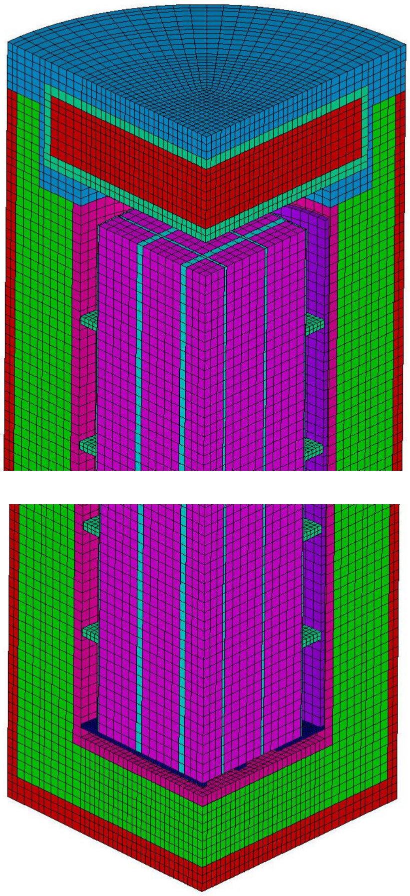
(b) Internal view.

Figure 1. Main parts of the cask model (1:2 scale).

Structural parts as lifting trunnions, bolts and threads were not modeled once these parts have little influence on the temperature distribution. In this figure one can identify the internal and the external cylinders with the shielding lead between them, including the bottom, the two flanges (the upper and the lower ones), the upper head with its shielding lead, on the lower flange, and the upper lid (connected to the upper flange). Inside there is the basket to accommodate the fuel elements which can be seen in details in figure 3.

The details of the inside structures can be seen in Figure 3.a to 3.c which shows the basket model where the main parts can be identified: the structure formed by the tubes with square section (no fuel elements) and their stiffening plates, the circular plate with holes at the bottom, the steel pieces in the corners and the lateral curved steel plate. The contacting walls among the square tubes and among tubes and the plates are considered as continuous material (this is a conservative hypothesis). All other contacting parts, even those welded ones, were modeled separately and a contact was defined between each pair. All surfaces in contact were modeled independently with a thermal contact conductance value defined with a specific Real Constant to identify each surface pair. Figure 3.c show more details of the basket with the relative positions of the parts in an upper view and the Figure 3.b shown the small circular plate that maintains the basket 10mm away from the internal cylinder bottom. Due to the double symmetry there are four of these pieces while in the design there are only three of them. This approximation is considered to be conservative once it allows more heat flow between the internal cylinder and the basket.

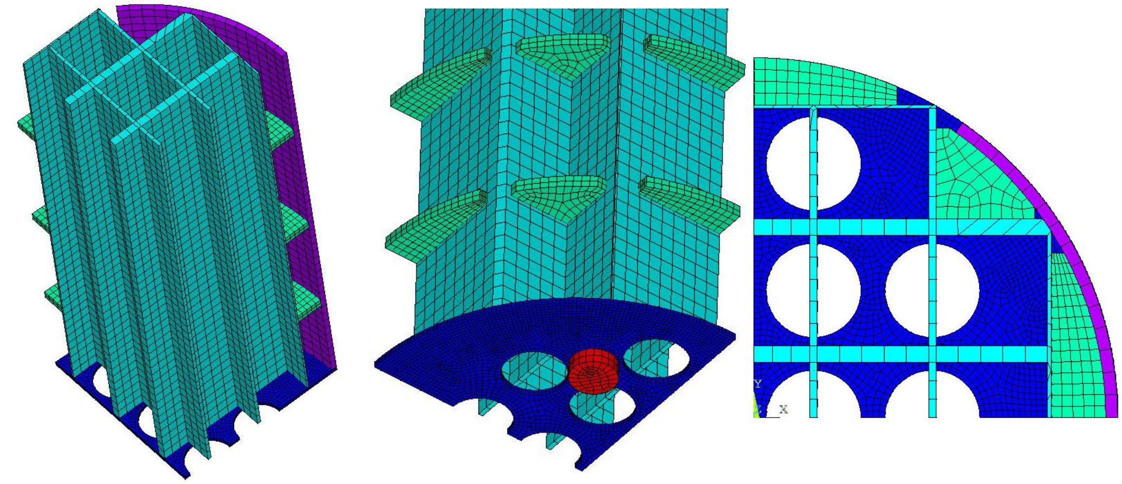
A contact can have up to 26 Real Constant values to characterize it. Two of them were defined in each contacting surface pair: the PINB and TCC (see next items). As an approximation, the same modeling hypothesis was adopted at each interface pair among the dummy fuel elements and the basket walls: there are two independent surfaces in each contact. Each dummy fuel elements to be used in the thermal test, as well as the actual ones, is formed by parallel aluminum plates. In the numerical model it is modeled as one block with an equivalent anisotropic material that has its thermal conductivity in the plate's plane greater than the conductivity perpendicular to the plates. The nominal gap between the fuel elements and the basket was ignored: the surfaces were modeled as in contact (again a conservative hypothesis). All other structural surfaces with some theoretical gap between them were modeled independently with no contact but with radiation and/or a convection coefficient between them.



(a) Upper part

(b) Lower part

Figure 2. Complete finite element model (90º).



(a) with lateral circular plate (b) bottom view (no lateral circular plate) (c) upper view

Figure 3. Finite element model of the basket (90º) – no fuel elements.

4. THERMAL ANALYSIS

The main materials properties adopted in the analyses are indicated in Table 1. Figure 4 shows the specific enthalpy X temperature curve [5] adopted for the shielding lead which changes phase, from solid to liquid at 327.1 ºC. The ANSYS program adjusts all input temperatures to the Kelvin scale by a defined offset value (273).

Table 1. Materials properties adopted in the analyses [5, 6, 7]

	Stainless Steel	Lead	Aluminum	units
E - Young's modulus	200e9	14e9	70e9	N/m^2
ν - Poisson's ratio	0,30	0,42	0.34	---
ρ - Density	7500	11500	2700	Kg/m^3
K_{xx} – Thermal conductivity	20.0	30.0	230	W/m.ºC
C – Specific heat	500	150	1000	J/Kg.ºC
Emissivity (all internal surfaces)	0.5	---	0.5	---

4.1. TCC – Thermal Contact Conductance

The temperature distribution at the interface between two bodies in contact is depicted in Figure 5 where the bodies are supposedly thermally insulated to allow the heat flow in one direction. The temperature drop is due to the so-called thermal contact resistance between the surfaces in contact and it depends on several factors. The main ones are the contact pressure and the surface roughness.

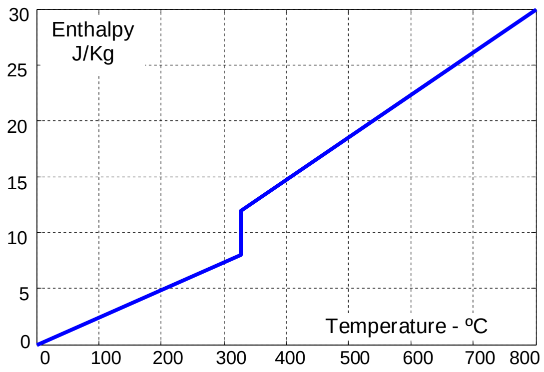


Figure 4. Specific Enthalpy X Temperature – Lead [5]

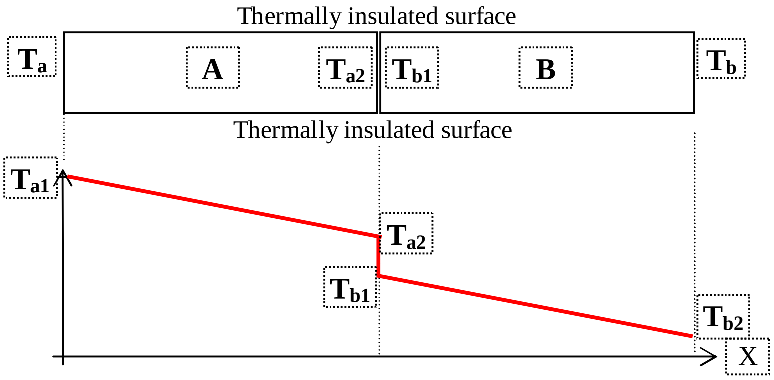


Figure 5. Temperature distribution at the interface between two bodies in contact.

The greater the contact pressure the greater the contact conductance or, in other words: the smaller the contact resistance. (In the following discussion it is assumed parallel surfaces, like in Fig. 5) As the in-contact surfaces roughness increases also the average distance among the surfaces growths increases, and so the thermal contact resistance. One should remember that in this situation the contact occurs at discrete points and the actual contact area is less than the common surface area. At the same time, interstitial material existing between and inside the surface roughness can increase the heat flow between the surfaces. More details can be seen in [6]. The ANSYS program uses the Thermal Contact Conductance, TCC, as the parameter associated with this phenomenon.

So, the thermal contact resistance between two surfaces is a very complicated phenomenon and measuring it is a very difficult task. Due to this fact, conservatively in the initial analyses performed in this work it was decided to consider all contact as perfect continuity which means with no thermal resistance. In the future, the thermal contact resistance will be calibrated using experimental results from the cask thermal test.

4.2. Thermal Contact – Elements and Parameters

The contacting surface pairs are defined by two element types: TARGE70 (3-D target segment) and CONTA174 (3-D 8-Node Surface-to-Surface Contact element), from the ANSYS library [4]. All contacts were defined with the TEMP (temperature) degree of freedom and with the 'always bonded' option and the option of contact detection on nodal point, using the appropriated KEYOPT values. The adopted contact algorithm is the 'Augmented Lagrangian' which is the default option.

In the ANSYS program, each contact pair is identified by a unique number named REAL CONSTANT, with a PINBALL parameter defined with a 3mm value for all contacts. This parameter tells the program the extension around a given node were it should search for contact. There are 31 contact pairs: 08 pairs among the basket structural parts, 12 pairs among lead shielding and internal & external cylinder walls and the upper head walls, 1 pair between basket and **i**nternal cylinder (bottom), and 10 pairs between basket and dummy fuel elements.

4.3. Thermal Radiation (Among Internal Surfaces)

Radiation is another heat transfer process when the heat energy 'travels' through the vacuum. Every body with a temperature above absolute zero (0 ºK or -273 ºC) radiates energy (heat). It depends on its emissivity and the rate at which the (heat) energy would radiate from them if they were a perfect black body. So, in this model, those surfaces closed to each other, but not physically in contact, and one in front of the other, can experience heat flow by radiation if an emissivity is defined among them as indicated in table 1. Each radiating surface pair is defined, in the program/model, by the same 'Real Constant' value with the associated parameters (basically the emissivity). In the present analysis only two internal surface pairs can change heat by radiation:

. the circular plate in the basked base and the bottom of the internal cylinder.
. the circular plate in the basket lateral side and the internal cylinder.

Among others available in the ANSYS program [4] to consider radiation, the 'Radiation Matrix Method' option was adopted. In this method, for each surface pair, a radiation matrix (like a superelement matrix) is created, using the ANSYS AUX12 routine, defining the geometry (3D), the emissivity, the Stefan-Boltzmann Constant (5.67e-8 W/m^2K^4), and the temperature offset (273 ºC).

In the (transient) analysis beginning these surfaces are cold and isothermal (20 ºC) and after some time there is a temperature difference among them and as this difference becomes greater the heat flow by thermal radiation among them becomes more important. This type of heat transfer (radiation) makes the analysis strongly non-linear which adds to the inherent material non-linearities due to the temperature (as the change phase of the lead and the thermal conductivity).

4.4. Thermal Convection

Convection is the heat transfer processes between a body surface and a fluid in motion. It depends, mainly, on the (bulk) fluid and the surface temperature difference as well as the fluid properties and velocity. The fluid velocity makes the convection ‘natural’ or ‘forced’. Other classification is done when considering the convection in a closed or in an open space (it depends on the relative dimensions of the involved body and space where the fluid flows). In the model under analysis there are both cases. There are specific experimental correlations to define the heat convection coefficient (h) for each situation. They are available in the literature as, for instance, in ref. [7].

Internal Surfaces. As it was done for the thermal radiation, those internal surfaces closed to each other, but not physically in contact, and one in front of the other, can exchange heat by thermal convection if a thermal (natural) convection coefficient is defined among them. Although the internal environment is at rest and the spaces are small and the influence of this kind of heat exchange is expected to be small it is considered in the developed model by defining appropriated heat convection coefficients (HF).

External Surfaces. In these preliminary analyses, heat convection coefficients (HF) were defined in the external cask surfaces to simulate the different situations during the test: isothermal (20 ºC) at the beginning, inside the oven at 800 ºC for 30 min and the cooling phase (simulated for only two hours) in an environment at 20 ºC under natural convection. As an approximation, and in a preliminary approach, the oven environment was simulated by a forced thermal convection coefficient value. The program needs two parameters: HF and TF (the bulk temperature, the fluid temperature far from the surface):

Heating phase (Inside the oven): HF = 800 ºC, TF = 100 W/m^2.ºC.
Cooling phase (outside the oven - air): HF = 20 ºC, TF = 30 W/m^2.ºC.

4.5. Symmetries

Due to the strong no-linearity of this analysis, about 60 load steps were defined. The option to simulate the thermal radiation oven environment by forced thermal convection was done towards a reduction in the cpu time spent in these preliminary analyses. In this same direction is the model size reduction by considering double symmetry (null heat flow) in the surfaces at X = 0 and Y = 0. The geometry is not precisely symmetrical, nor does the presence/ consideration of thermal radiation allow this, however this model option seems reasonable once these are exploratory analyses. Once the model is considered as working well the final material properties will be input, the oven environment will be simulated by radiation and the model will be doubled to allow the analyses whose results will be compared with the experimental ones.

5. PRELIMINARY RESULTS

Figure 6 shows the temperature distribution along 1h (out of the 2h analysis) at two points in the lateral lead between the cylinders. Both points are at the same level (about half height of the lead), one of them at about $L_t/3$ and the other about $2L_t/3$ inside the lead (L_t being the lead thickness).

Figure 7 and 8 show the temperature distributions respectively at 30min (the end of the heating period) an d at 50min (or: just after 20min of the cooling period): (a) at the full model, (b) only at the basket (no fuel elements) and (c) in a bottom view of the basket with no circular plate where one can notice the hot spot due to the connection between the basket and the internal cylinder by a circular piece.

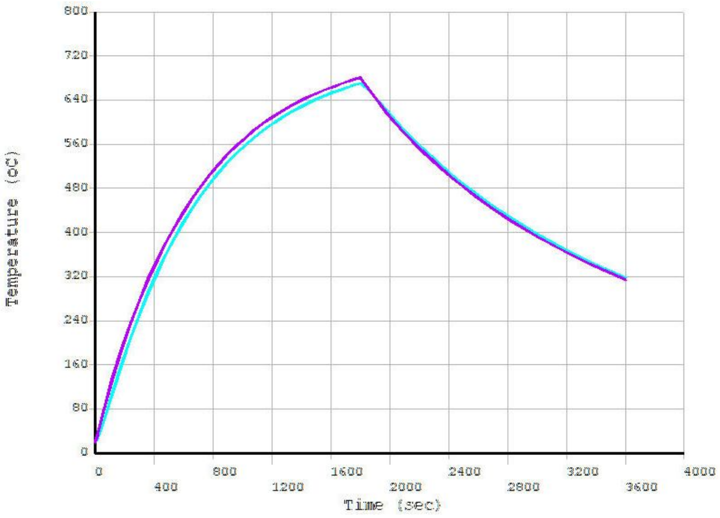
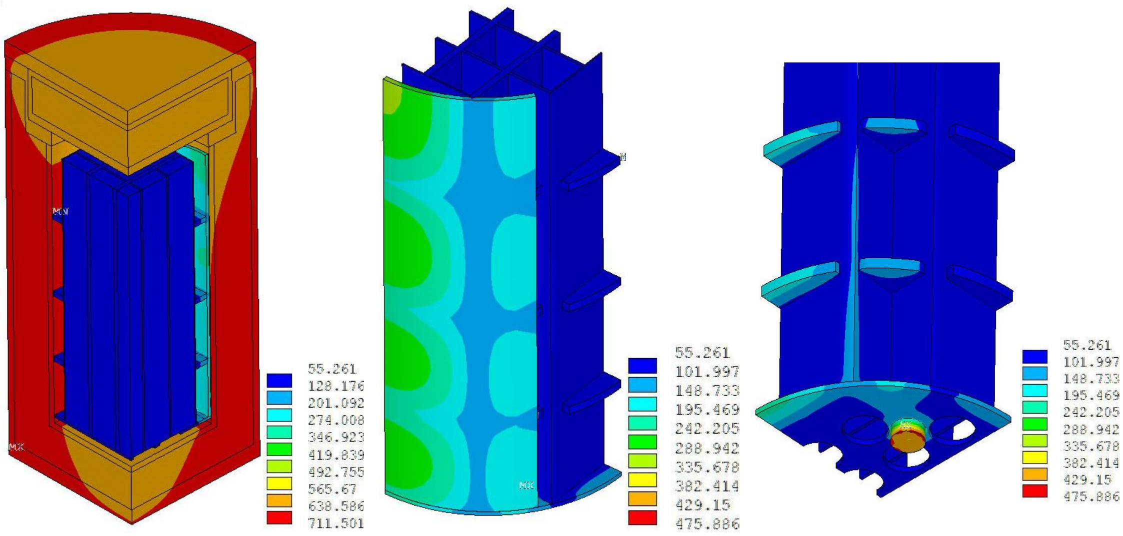


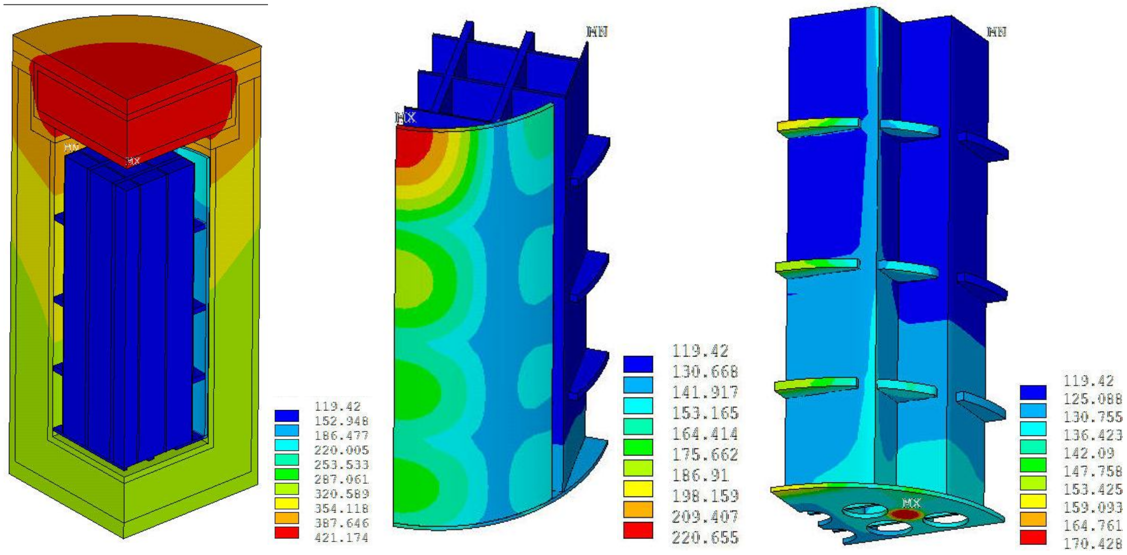
Figure 6. Temperature distribution in the lead (at two points) – along the 1h analysis.



(a) full model **(b) basket (no fuel elements)** **(c) basket (bottom) (with the circular piece)**

Figure 7. Temperature distribution at 30min (end of the heating period).

In Figure 8.c the circular piece was removed but, still, there is a hot spot where it should be.



(a) full model **(b) basket (no fuel elements)** **(c) basket (bottom) (no circular piece)**

Figure 8. Temperature distribution at 50min (after 20min of cooling).

6. CONCLUSIONS

This paper showed the chosen approach to the numerical simulation of the thermal test of a 1:2 scale model of a dual purpose cask (transportation and/or storage) for spent fuel elements from nuclear research reactors and a brief description of the cask model was also presented. The numerical model using finite elements and the hypothesis to build it were detailed and discussed.

As the main purpose of the paper is to present the proposed approach for the thermal test numerical simulation, only some preliminary numerical results are showed without any comparison to the experimental ones. With the need of some calibration in the model and material properties, as the thermal contact conductance, to predict the prototype field tests results some comparisons between the numerical results and the experimental ones for scale model tests will be conducted in near future.

REFERENCES

1. CNEN, “Transporte de Materiais Radioativos”, Comissão Nacional de Energia Nuclear, Rio de Janeiro, RJ (1988) (CNEN-NE-5.01)
2. M. Mattar Neto, C. A. J. Miranda, G. Fainer, Avaliação do projeto de amortecedores de impacto para embalagens de transporte de elementos combustíveis irradiados de reatores nucleares de pesquisa. CONEM 2008 - V NATIONAL CONGRESS OF MECHANICAL ENGINEERING, August 25 – 28, 2008 - Salvador – Bahia – Brazil. (2008)
3. M. Mattar Neto, C. A. J. Miranda, G. Fainer, R. P. Mourão. Analytical and Numerical Evaluation of the Impact Limiters Design of a Research Reactors Spent Fuel Transportation Package Half Scale Model under 9 m Drop Tests. In: Proceedings of PVP2008, 2008 ASME Pressure Vessels and Piping Division Conference, July 27-31,

2008, Chicago, Illinois, USA. (2008)

4. ANSYS v11.0 ANSYS Structural Analysis Guide & Theory Reference Manual.(2008)
5. Handbook on Lead-Bismuth Eutectic Alloy and Lead Properties, Materials Compatibility, Thermal-Hydraulics and Technologies. NEA – Nuclear energy Agency, 2007 Edition. http://www.nea.fr/html/science/reports/2007/nea6195-handbook.html (2007)
6. Fletcher, L. S., "Recent Developments in Contact Conductance Heat Transfer". Journal of Heat Transfer, November 1988.
7. Frank P. Incropera, David P. DeWitt, Fundamentals of Heat And Mass Transfer, 5th ed., John Willey Sons, Inc., 2002.