A THERMAL ANALYSIS OF A SPENT FUEL TRANSPORTATION CASK UNDER SEVERE FIRE ACCIDENT CONDITIONS

Arivaldo Morais do Sacramento, ams@cdtn.br
Vanderley de Vasconcelos, vasconv@cdtn.br
Centro de Desenvolvimento da Tecnologia Nuclear – CDTN/CNEN
Caixa Postal 941
30123-970, Belo Horizonte, MG, Brazil.

Elizabete Jordão, bete@feq.unicamp.br
Universidade Estadual de Campinas – UNICAMP
Caixa Postal 6066
13083-970, Campinas, SP, Brazil.

Abstract. *All casks that contain radioactive materials should satisfy a 30 min Hypothetic Accident Conditions – HAC thermal test under 800°C. There are two approaches for evaluating the safety implications of a cask engulfed in a fire: the first is by experimental investigation and the second by numerical modelling. An issue of significant concern in nuclear area is the capability of compute codes to model pertinent physical phenomena without requiring extensive use of computer resources. In this work, specific conditions were imposed on two different cask configurations to evaluate the ability of a popular heat-transfer code, specifically, ANSYS code, to simulate a realistic physical problem. On the first cask the ability of the code to simulate conduction and convective boundary condition was tested; on the second one, exchange heat by thermal radiation was simulated. A comparison between some experimental data of 800°C heat condition and numerical results are presented.*

Keywords*: cask, fire, thermal analysis, hypothetical accident condition, ANSYS*

1. INTRODUCTION

All casks that contain radioactive materials must be evaluated in order to determine their expected normal operating temperatures and their responses to the accident conditions specified in the regulations. The main conditions the cask should satisfy to be qualified are: a Hypothetic Accident Conditions – HAC 9 m drop tests, a 30 min HCA thermal test under 800°C and a variety of leak tests, including 10 m immersion tests.

This work will focus on the thermal analysis. There are two approaches for evaluating the safety implications of a cask engulfed in a fire: the first is by experimental investigation and the second by numerical modeling. An issue of significant concern in nuclear area is the capability of computer codes to model pertinent physical phenomena without requiring extensive use of computer resources. In this work, specific conditions were imposed on two different cask configurations to evaluate the ability of a popular heat-transfer code, the ANSYS code, to simulate realistic a physical problem.

2. THERMAL PROBLEMS

The NRC regulations, given in 10 CFR Part 71, define a thermal event as " exposure of the hole specimen for not less than 30 minutes to a heat flux not less than that of a radiation environment of 800 °C with a emissivity coefficient of at least 0.9. For purposes of calculation, the absorptivity must be either that value which the cask may be expected to possess if exposed to a fire or 0.8, whichever is greater".

To be modeled completely the thermal event requires simulating of conduction, convection, and radiation (Glass, 2001); the problems included in this work will exercise these facets of the code. Additional phenomena, such as phase change and boiling-water heat transfer, are not included in this problem set.

2.1. Model A

Model A, as shown in Fig. 1, is a two-region cylinder. The interior region – Region I – contains a volumetric heat source representative of the internal decay load from irradiated fuel. This problem, which has a closed-form analytical solution, tests the ability of the code to simulate both conduction and convective boundary condition.

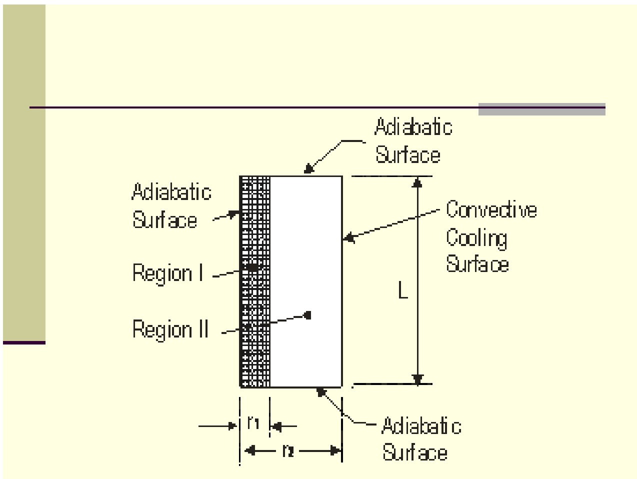

Figure 1. Model A – cylinder with internal heat generation (Glass, 2001)

Both the ambient environment and initial cask temperature are 54.4 °C. The geometric and thermal characteristics of Model A are given in Tab. 1

Table 1. Model A – Cylinder with internal heat generation: characteristics (Glass, 2001)

Characteristics	Region I	Region II
Radius, r	$r_1 = 27.43$ cm	$r_2 = 91.44$ cm
Length, L	457.2 cm	457.2 cm
Density, ρ	16.03 kg/m^3	16.03 kg/m^3
Specific heat, c_p	4.18×10^3 J/kg K	4.18×10^3 J/kg K
Conductivity, k	69.2 W/m K	69.2 W/m K
Heat source, Q	11.09 kW/m^3	11.09 kW/m^3
Convective coefficient, h_c		5.67×10^{-3} kW/m^2 °C

2.2. Model B

Model B, which is shown in Fig. 2, is based on a prototypic cask configuration consisting of several different annular regions. Region I contains a volumetric heat source simulating irradiated fuel decay heat. Region II is a monolithic cask wall, Region III is a voided neutron shield, and Region IV is the thermal radiation shield. Regions II and IV heat solely by thermal radiation. Heat is also exchanged with the surrounding environment from Region IV by thermal radiation. For simplicity, all surfaces and environment are assumed to be black.

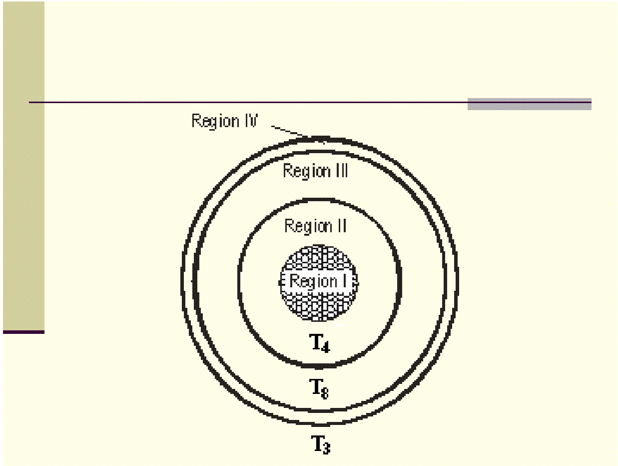

Figure 2. Model B – cask with annular regions (Glass, 2001)

The following problem simulates exposure of Model B to the regulatory fire and involves two analyses. First, a steady-state solution, with radiant heat loss to a 54.4 °C environment must be calculated; second, using the steady-state solution for the initial conditions, the fire is simulated by an increasing in the ambient temperature to 800°C during 30 minutes

The geometric and thermal characteristics for Model B are given in Tab. 2.

Table 2. Model B – Cask with annular regions: characteristics (Glass, 2001)

Characteristics	Region I	Region II	Region III	Region IV
Radius, r	16.51 cm	38.74 cm	53.98 cm	54.61 cm
Density, ρ	2707 kg/m^3	7832.8 kg/m^3		7832.8 kg/m^3
Specific heat, c_p	0.89×10^3 J/kg K	0.47×10^3 J/kg K		0.47×10^3 J/kg K
Conductivity, k	242 W/m K	45 W/m K		45 W/m K
Heat source, Q	38.32 kW/m^3			

3. ANALYSIS RESULT

3.1. Model A

In order to simplify the modeling, a strategic decision was taken to consider the problem as two dimensional; due to the symmetry of the problem, the cask model was based on a half section through the diameter positioned midway along the length of the cask. The symmetry and convective boundary conditions of the cask are shown in Fig. 3. The model was discretized with 100 2-D elements plane 77 (Madenci and Guven, 2006); this element has eight nodes with a single degree of freedom, temperature, at each node. Convection was entered as a surface load at the element faces; heat generation rate was entered as element body load at the area one.

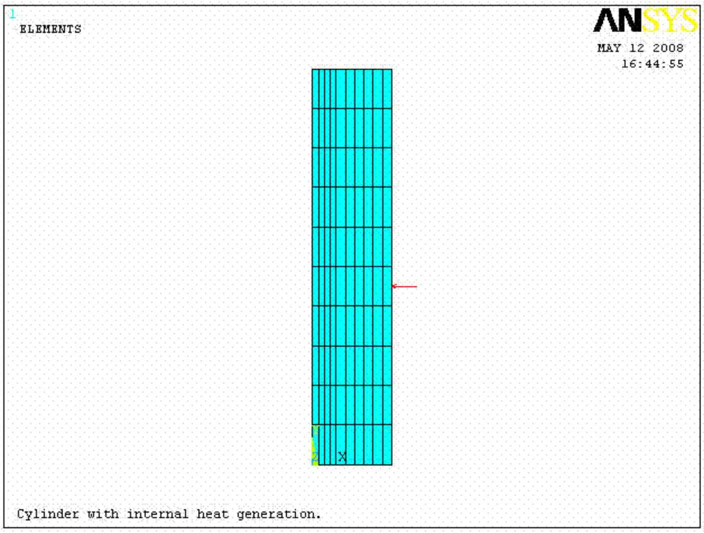

Figura 3. Symmetry and convective boundary conditions

The temperature contour of the cask is shown in Fig. 4; the peak steady state temperature attained at the cask centerline is determined by heat generation rate and the thermal conductivity of the material.

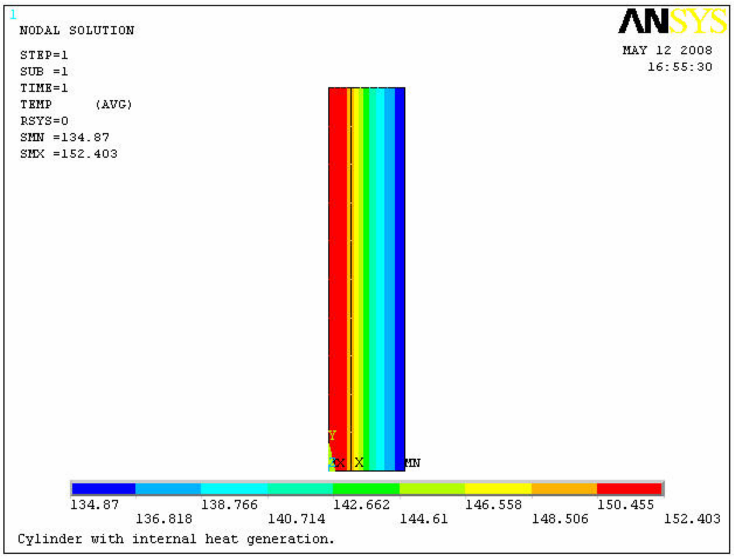

Figure 4. Temperature distribution contour of the cask simulated with ANSYS

The comparison of the numerical results, using finite element program ANSYS, and the expected results revealed a very good agreement for this case; the results are given in Tab 3. below.

Table 3. Model A – Cylinder with internal heat source: results of simulations using ANSYS

Solutions	Centerline (^{o}C)	Interface (^{o}C)	Outer edge (^{o}C)
Exact closed form	152.0	149.0	135.0
Numerical	152.4	149.4	134.9

3.2. Model B

This problem tests the ability of the code to simulate conduction and thermal radiation (Maxa, 2003). PLANE55 was used as plane element for thermal conduction capability; the element has four nodes with a single degree of freedom, temperature, at each node. For radiation analysis, the radiation link element LINK31 was used. This two-node element is used between nodes and requires specification of area, form factor, and emissivity (Holman, 2002) as real constants. Heat generation rate was entered as element body load at the element one. The model was discretized using cylindrical coordinate system; it was used seven elements and 11 nodes. The finite element grid is shown in the following Fig. 5.

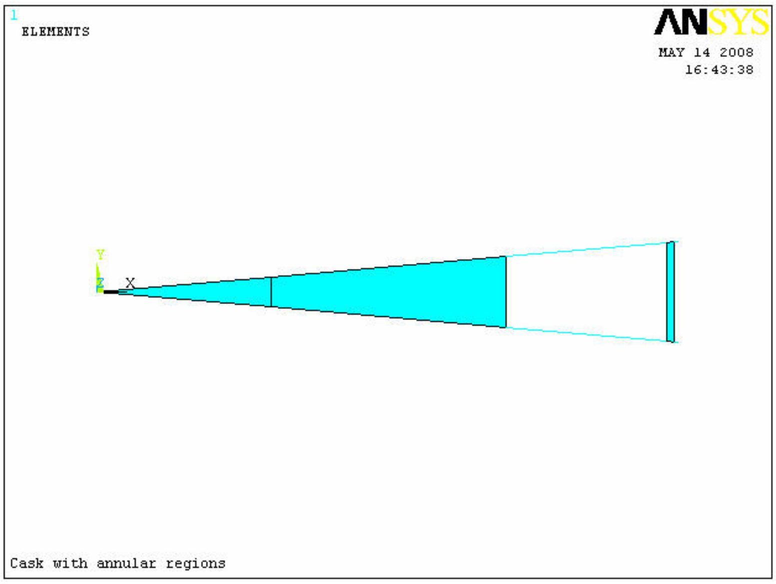

Figure 5. Domains for thermal analysis for the cask simulated with ANSYS

The temperature distribution of the cask is shown in Fig. 6; 213.6 °C is the temperature at the interface between Regions I and II; 204.3 °C is the temperature at the interface between Regions II and III, and 136.6 °C is the temperature at the outer surface. These calculated steady-state temperatures are used as the initial conditions prior to the start of the fire accident simulation.

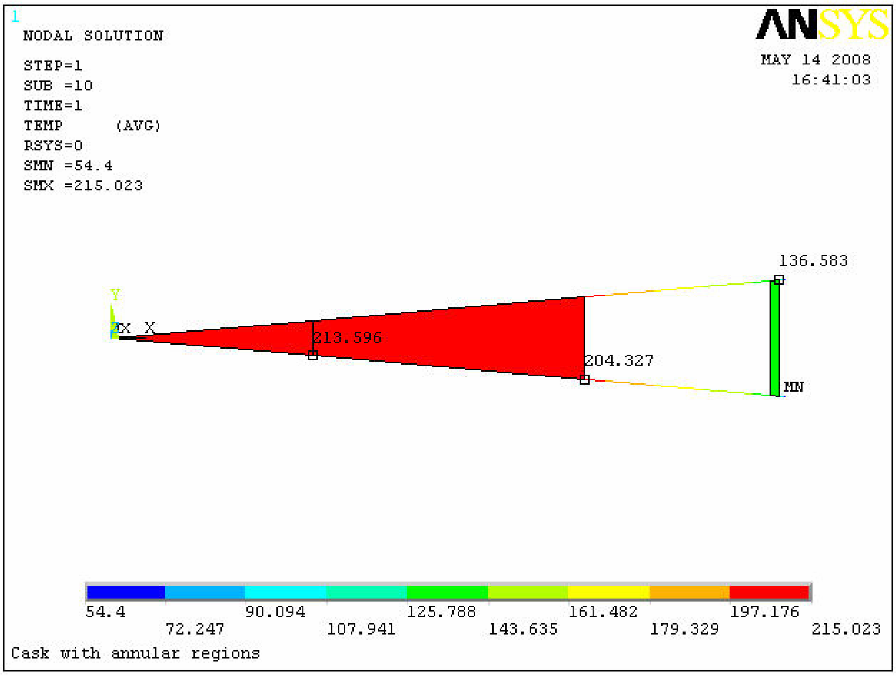

Figure 6. Temperature distribution of the cask simulated with ANSYS

The comparison of the numerical results, using finite element program ANSYS, and the expected results, revealed also a very good agreement for the case involving conduction and thermal radiation; the results are given in the following Tab.4.

Table 4. Model B – Cask with annular regions: steady-state case results

Position	TARGET Temperature (°C)	ANSYS Temperature (°C)	RATIO
T_4	214.0	213.6	1.00
T_8	204.0	204.3	1.00
T_3	137.0	136.6	1.00

For the transient case, the temperature range at the outer surface of the cask is shown in Fig.7 bellow; the general trend of the temperatures in the model is similar to that found experimentally.

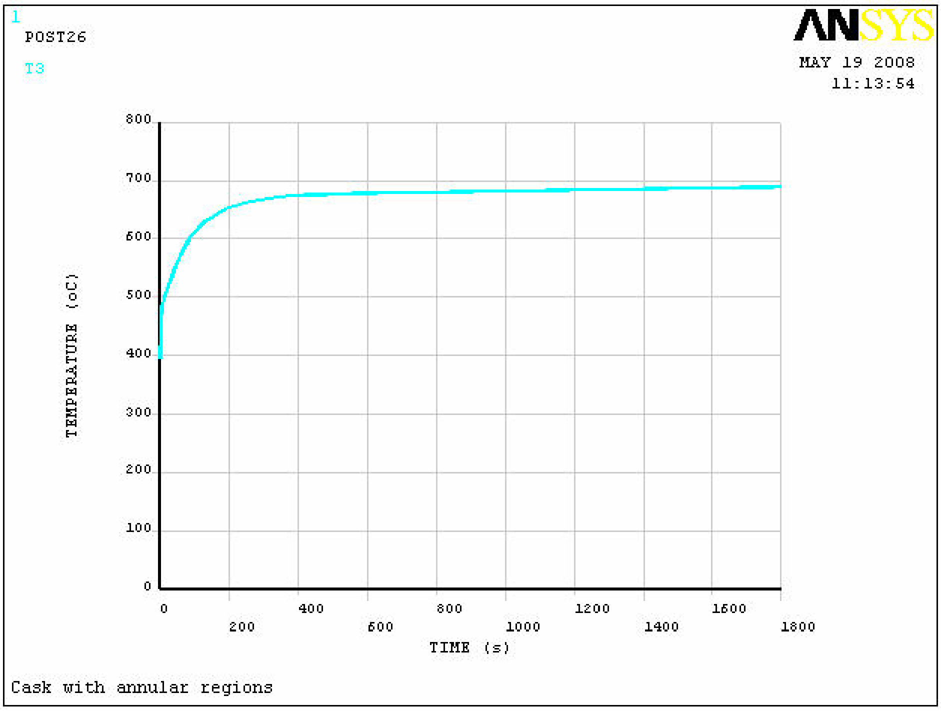

Figure 7. Temperature T_3, at cask surface versus time, simulated with ANSYS

The temperature distribution of the cask, for the transient case, is shown in the following Fig. 8; 274.4 °C is the temperature at the interface between Regions I and II; 375.7 °C is the temperature at the interface between Regions II and III, and 689.3 °C is the temperature at the outer surface.

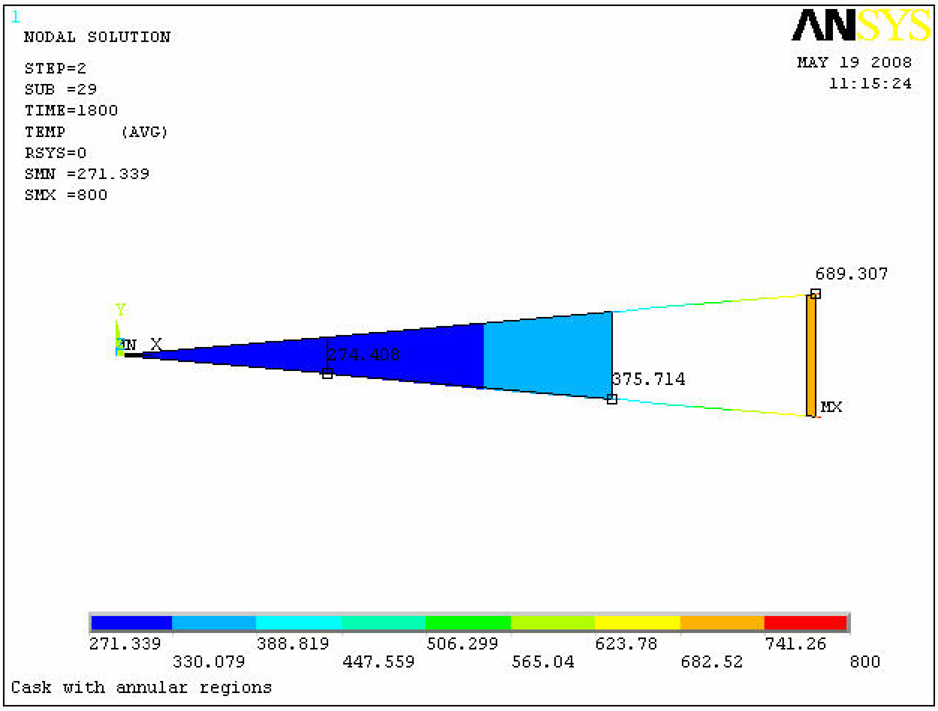

Figure 8. Temperature distribution of the cask simulated with ANSYS

The comparison of the numerical results using finite element program ANSYS and the expected results revealed also a very good agreement for the transient case; the results are given in Tab.5 below.

Table 5. Model B – Cask with annular regions: transient case results, time = 30 min.

Position	TARGET Temperature (^oC)	ANSYS Temperature (^oC)	RATIO
T_4	263.0	274.4	1.04
T_8	376.0	375.7	1.00
T_3	689.0	689.3	1.00

4. CONCLUSION

The purpose of this study was to create a computer model capable of predicting the thermal behaviour of a cask involved in a fire. To achieve this goal a 2D model, using the finite element code ANSYS, was developed. Two casks with different configurations were analyzed: Model A and Model B. For the first model, the ability of the code to simulate conduction and convective boundary conditions was tested; the calculated temperature at selected points in the integration domain revealed a very good agreement with the expected results. For the second model, the peak centre temperature under steady-state conditions is determined by the heat generation rate and the thermal conductivity of the cask; during the 30 min of the fire, the change in center temperature is relatively small. However, the outer surface responds rapidly to exposure to the fire. The analysis of the results obtained by the present work shown that ANSYS code presents sufficient ability to simulate thermal problems involving conduction, convection and thermal radiation.

.

5. ACKNOWLEDGEMENTS

The authors would like to express their gratitude to Center of Development of Nuclear Technology – CDTN/CNEN that sponsored this work, and their colleagues of CDTN/CNEN and State University of Campinas – UNICAMP who were involved with this work.

6. REFERENCES

Glass, R.E., 2001, "Appendix B: Thermal Code Benchmark Problems", Sandia National Laboratories, DE-AC04-76DP00789, Albuquerque, New Mexico, pp. 1-11

Holman, J.P., 2002, "Heat Transfer", McGraw Hill, New York, USA, 666 p.

Madenci E. and Guven I., 2006, "The Finite Element Method and Applications in Engineering Using ANSYS", Springer, Arizona, USA, 686 p.

Maxa, Z., 2003, "Influence of Radiation in Heat Distribution Model of Aluminium Foam. Single Cell Simulations" Laser Zentrum Hannover, Hannover, Germany, 12 p.

5. RESPONSIBILITY NOTICE

The authors are the only responsible for the printed material included in this paper.