Cracking Propagation in Concrete Bloc Exposed to Cryogenic Temperature

L.BOUCHELIL, L.DAHMANI

Abstract**—** In order to investigate the cracking propagation in the concrete bloc exposed to low temperature, The ANSYS finite element code has been employed for performing a sequential, non linear, transient thermal-structural analysis, taking into account the thermal dependent properties of the concrete as thermal conductivity and specific heat. Temperature distribution data of thermal analysis is required in the coupled field analysis finally to obtain and analyze thermal stresses.

An original concept based on three dimensional stress states of the integrations points in each finite element is described. The global relative crack density (GRCD) is suggested to denote the cracking state of the entire concrete bloc, which may serve as an appropriate index to evaluate the overall deterioration level of the structure.

Keywords**—** Ansys model, Crack propagation, Concrete bloc, Cryogenic temperature.

I. INTRODUCTION

LNG (Liquefied Natural Gas) has the cryogenic temperature of –160°C to ensure the minimum storage volume when stored in LNG tank [1,2]. Among various types of LNG storage tanks, the full containment above-ground type with a double safety system: outer concrete tank; and inner steel tank. Normally, the inner tank contains LNG, but when the LNG leaks from the inner tank, the outer concrete tank comes into contact with LNG. Under this accidental case, it is indispensable for the outer tank to keep the liquid tightness in order to safely contain the LNG before taking any countermeasure. It is, therefore, proposed to take up a heat conduction problem using finite element method with ANSYS software to obtain temperature distribution data of a concrete tank at cryogenic temperatures.

The basis for thermal analysis in ANSYS [3,4] is a heat balance equation obtained from the principle of conservation of energy. (For details, consult the *ANSYS, Inc. Theory Reference*.) The finite element solution performed via ANSYS calculates nodal temperatures, and then uses the nodal temperature to obtain other thermal quantities.

The elastic stresses, induced by mechanical constraints and thermal strains resulting from the previous analysis, have been calculated.

II. FINITE ELEMENT MODELING

A. Physical model

A solid concrete of 1.0 m long, 1.0 m wide, and 0.2 m thick shown in **Fig.1** is descretized with a 3D finite element model as shown in **Fig.2**. Its mechanical properties are given in **Table 1**.

A thermal version of the model was used to calculate the temperature profile in the concrete bloc; a structural version of the model then read the temperature profile to calculate stresses.

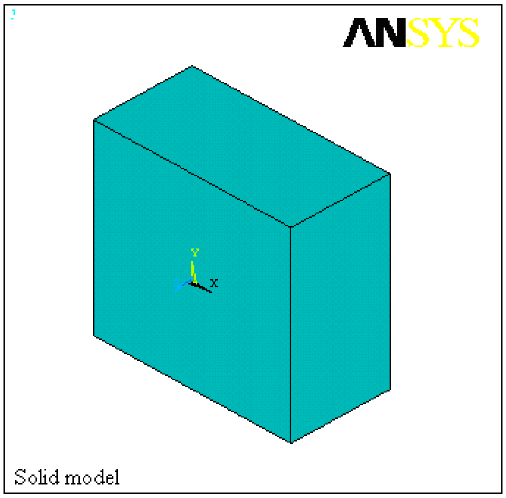

Fig. 1: Solid model (3-D)

A. Reinforced Concrete

A three-dimensional eight nodded solid element having thermal degree-of-freedom (element type solid 70 in ANSYS 15.0) is chosen for heat conduction problem (Fig.3).

The distributions of thermal elastic stress components were then calculated by switching the SOLID 70 thermal element to SOLID 65 structural element **(Table 2)** which is used for 3-D modelling of solid structures.

Lahlou DAHMANI is with the Civil Engineering Department, University Mouloud Mammeri, Tizi-Ouzou , Algeria (e-mail: lahlou_d@yahoo.fr)

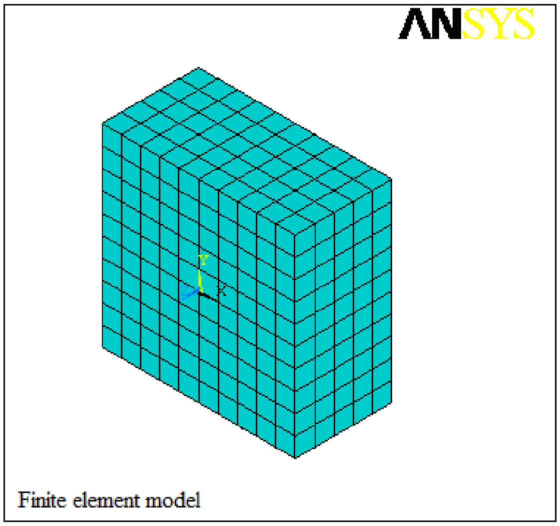

Fig. 2: Finite element model (3-D)

Table 1: Material properties of concrete [1]

Material properties	**values adopted**
Compressive strength	$f'_c = 30Mpa$
Tensile strength	$f_t = 3Mpa$
Strain at peak stress	$\varepsilon_0 = 0.002$
Ultimate strain	$\varepsilon_{cu} = 0.003$
Elastic modulus	$E_c = 30Gpa$
Poisson's ratio	$\nu = 0.2$
Density	$\rho = 2400kg/m^2$
Heat transfer coefficient	$h = 50W/m^2 °C$
Specific heat capacity	$c = 1000j/kg.°C$
Thermal expansion coefficient	$\alpha = 1.0 \times 10^{-5} /°C$

Table 2: Thermal structural model

Element	**Thermal**	**Structural**
Type	Solid 70	Solid 65
Number of nodes	8	8
Number of DOF per node	1	3
Nature	Temperature	Displacement Ux , Uy and Uz

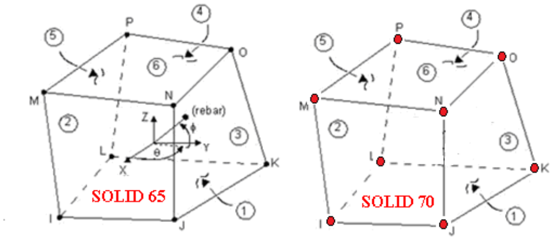

Fig.3: Thermal structural model

B. Transient thermal analysis

A *transient thermal analysis* determines the temperature distribution and other thermal quantities under conditions that vary over a period of time.Typical thermal quantities of interest are:

- The temperature distributions,
- The amount of heat lost or gained,
- Thermal gradients,
- Thermal fluxes.

The mathematical solution for the element's conduction heat transfer is based on the first law of thermodynamics - energy conservation law [5,6].

$$div[k.grad(T)] + Q'(T) = \rho c \frac{\partial T}{\partial t} \qquad (1)$$

.

where ρ is the density of material, c is the heat capacity, and k is the thermal conductivities of the concrete tank varying with temperature.

Based on differential equation (1) with tacking into account of the spatial temporal boundaries conditions, the heat balance for the structural nodes at time (t+Δt) is given by:

$$[C]\{\dot{T}\} + [K]\{T\} = \{F\}. \qquad (2)$$

$[C]$ heat capacity matrix c

$[K]$ conductance matrix containing the thermal conductivity terms(k) and heat exchange coefficients (α).

$\{\dot{T}\}$ nodal temperature rate vector $\partial T / \partial t$

$\{F\}$ thermal load vector (temperature ect…)

A transient thermal analysis follows basically the same procedures as a steady-state thermal analysis. The main difference is that most applied loads in a transient analysis are functions of time. To specify time-dependent loads, one can divide the load-versus-time curve into load steps.

The temperature is obtained via Galerkin finite element technique as implemented by ANSYS software package [4,7,6].

C. The time integration parameter

The time integration parameter θ relates temperature difference to temperature rate:

$$T_{n+1} - T_n = \Delta t_n (1-\theta)\dot{T}_n + \Delta t_n \theta \dot{T}_{n-1} \ . \qquad (3)$$

Any value within $\frac{1}{2} \leq \theta \leq 1$ is unconditionally stable. That is, all solutions are stable regardless of how large a time step Δt_n is chosen. In Ansys the default setting is $\theta = \frac{1}{2}$, known as the Crank-Nickolson technique. It is usable in the majority of transient problems. Details about the algorithm are found in references [6].

III. CRACKING RESPONSE

Most deterioration processes are able to alter the porosity and permeability of concrete, trigger the initiation and growth of cracks, and thus impair the integrity of the concrete structure [2,8].

Ngo and Scordelis [9] first introduced the effect of cracking into the finite element analysis of reinforced concrete structures. There usually exist two finite element-based approaches to simulate the concrete cracking — discrete cracking approach and smeared cracking approach.

A. Discrete cracking approach

Discrete cracks are treated directly as geometric discontinuities, where the intact portion of concrete is generally assumed to behave elastically, while the crack propagation is modelled by changing the topology of the finite element model. Adaptive remeshing is required to implement the simulation. It is extremely difficult to implement the simulation of concrete cracking caused by the multiple coupled deterioration processes using the discrete cracking approach.

B. Smeared cracking approach

Smeared cracks are assumed to be spatially distributed over the entire volume represented by a local finite element, or only by the volume attached to one integration point within the finite element. The stiffness matrix of each concrete element is modified accordingly to accommodate the mechanical deterioration due to cracking without changing the topology of finite element model. This approach is used in our work and proves efficient when incorporated in finite element analysis.

The William and Warnke failure criterion [10] under multi-axial stress state is adopted to assess the initiation of failure and identify the corresponding failure modes (including cracking and crushing) at the centroid of a concrete element or one of its integration points (Fig.4).

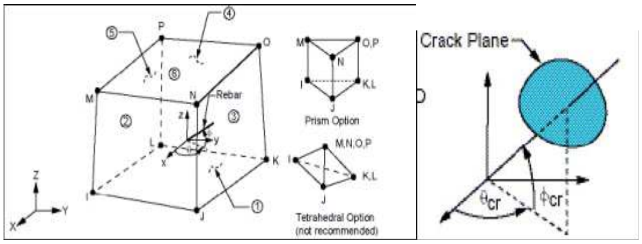

Fig. 4: SOLID65 concrete element

IV. NUMERICAL EXAMPLE

A plain concrete block 1.0 m long, 1.0 m wide, and 0.20 m thick is selected herein to illustrate the simulation procedure of crack propagation for concrete exposed to low temperature.

The total time length is assumed to be 48 hours, which is divided into 48 time steps with an average of 60 substeps for each time step. The average duration of each substep (i.e., about one minute is small enough to guarantee a stable solution. The finite element model is shown in Fig.5

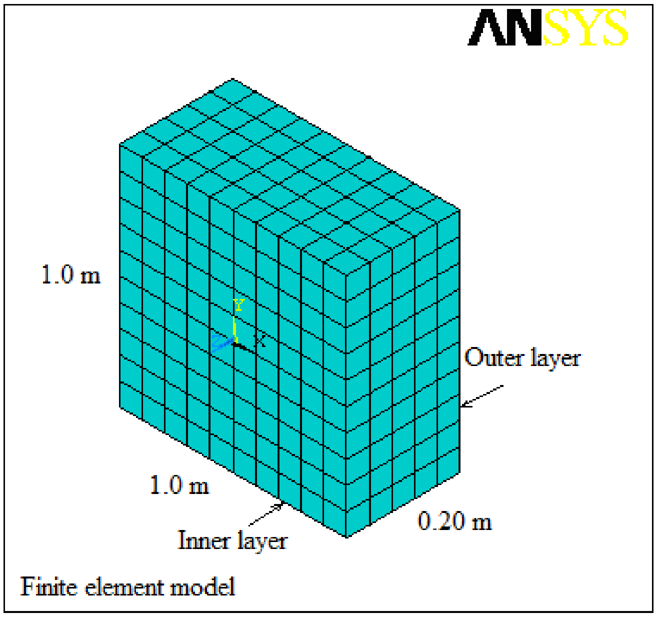

Fig.5: Finite element model

The temperature of -160°C is enforced on the inner layer of the plain concrete block. On the outer layer, a temperature of 50°C is applied. The initial temperature of the entire concrete block is assumed to be 20°C. Meanwhile, the four side surfaces of the concrete block are assumed to be isolated from any heat flux. In addition, structural displacements in all three orthogonal directions are restrained on the four side surfaces as structural boundary conditions.

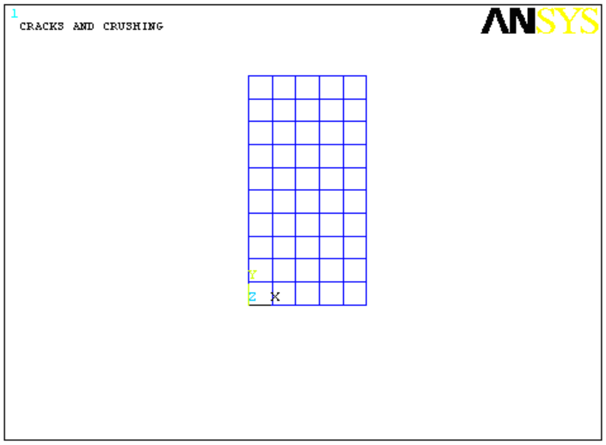

Fig. 6: Cracking state at (t = 10h)

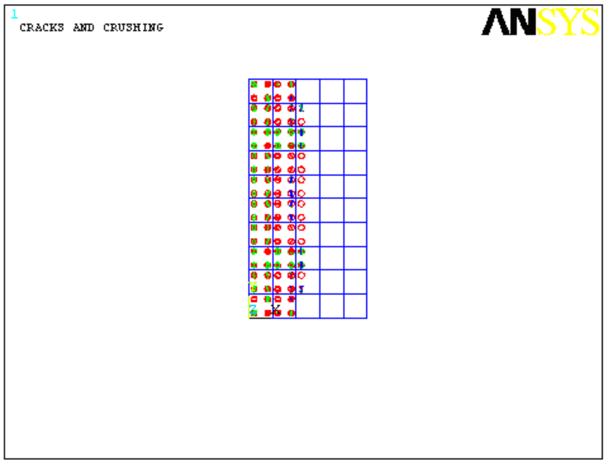

Fig. 7: Cracking state at (t = 24h)

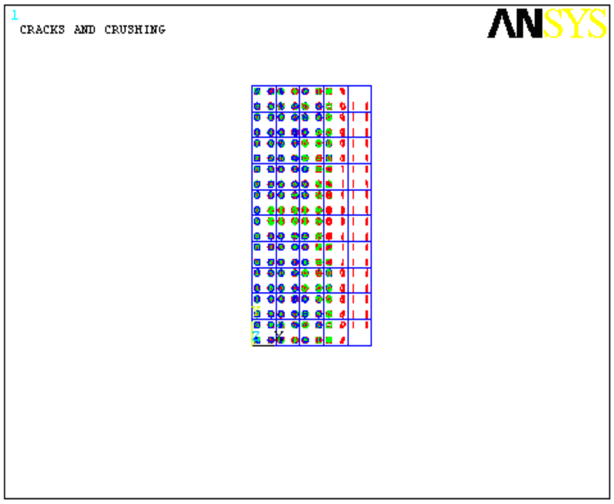

Fig. 8: Cracking state at (t = 48h)

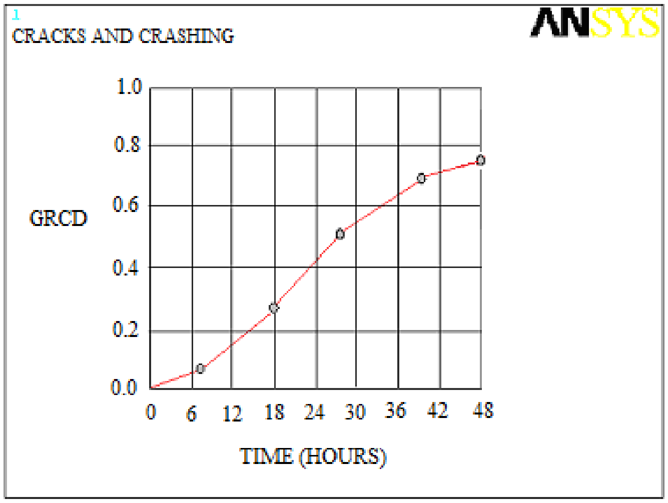

Fig.9: GRCD of concrete bloc

V. RESULTS AND DISCUSSIONS

The cracking state of the entire concrete block at each time step is illustrated in Figures 6to8, where the right side view of the plain concrete block in Fig. 5 is shown. It can be observed that more and more dense microcracks uniformly distributed over the entire concrete volume propagate with time.

The corresponding global relative crack density (GRCD) is shown in Fig.9, which increases with time and verifies the observation in Figures (6to8). It can be observed that about 75% of concrete finite elements are cracked, though each element may attain different cracking level.

It is observed that the values of GRCD (Fig.9) can be divided roughly into three phases. During the first phase (0–10 hours), almost no crack occurs, and GRCD approaches zero.

During the second phase (10-24hrs), a great number of cracks are generated, and GRCD increases quickly. During the third phase (24-48hrs), GRCD gradually approaches its maximum value, which may be explained as follows. The cracking state of the concrete element has reached to such an extent that its degraded stiffness does not allow it to carry more loads (the other uncracked or less cracked concrete elements may take the additional load)

VI. CONCLUSION

An integrated finite element-based computational framework has been developed in this paper to simulate the cracking propagation in concrete bloc exposed to low temperature.

The smeared cracking approach is utilized to simulate the crack propagation.

Finally, the global relative crack density (GRCD) is suggested to denote the cracking state of the entire structure and serve as an appropriate index to evaluate the overall deterioration level of the structure. This methodology establishes a reasonable basis for quantitative long-term durability assessment of concrete structures under coupled deterioration processes.

Future work can extend this methodology to consider more deterioration processes, including carbonation process, chloride penetration process, reinforcement corrosion and rust expansion process.

REFERENCES

[1] Dahmani, L.; Khennane, A. ; Salah, K. ; (2007). ; "Behavior of the reinforced concrete to cryogenic temperature", Cryogenics, Vol. 47, p. 517

[2] Dahmani, L., "Thermo mechanical response of LNG concrete tank to cryogenic temperatures", *Strength of Materials, Vol. 43, No. 5*, 2011*,* pp. 526-53.

[3] «ANSYS Theory Reference: Analysis tools", *001099, 9th ed. SAS IP, Inc.*

[4] ANSYS 15. "The general purpose of finite element software". Documentation.

[5] Chavan, R.; (1998-1999); "A thermomechanical analysis of the central column tiles", CRPP / EPFL - Lausanne, Internal report INT 195/99.

[6] Lienhard IV, J.H. and Lienhard V, J.H.; (2008) : "*A Heat Transfer Book*", (Phlogiston Press, USA)

[7] Madenci, Erdogan and Guvin, Ibrahim, " The finite Element Method and Applications in Engineering Using ANSYS", Springer, New York. (2006)

[8] Dahmani, L. , Khennane, A., Kaci,S. , "Crack identification in reinforced concrete beams using ANSYS software " , Strength of Material Journal, Ed. Springer New York , Vol.42,N°2, Mai 2010, pp. 232-240.

[9] Ngo, D. and Scordelis, A.C.; (1967); "Finite Element Analysis of Reinforced-Concrete Beams". *Journal of the American Concrete Institute*, Vol. 65, No. 9, pp.757-766.

[10] William, K.J. and Warnke, E.P.; (1975); "Constitutive Model for the Triaxial Behavior of Concrete". *Proceedings of the International Association for Bridge and Structural Engineering*, Vol. 19, ISMES, Bergamo, Italy, pp. 174.