Kufa Journal of Engineering (K.J.E)
ISSN 2207-5528
Vol. 5, Issue 1, Dec., 2013,P.P.127-140
Printed in Iraq

FINITE ELEMENT MODELLING OF CONCRETE SHRINKAGE CRACKING IN WALLS

Lecturer Dr. Ali N. Attiyah
Department of Civil
College of Engineering
University of Kufa

Prof. Dr. Hans Gesund
Department of Civil
College of Engineering
University of Kentucky

Mr. Mohammed H. Rasool
Department of Civil
College of Engineering
University of Kufa

Received: 6 / 3 / 2012; Accepted: 16/6/2013

Abstract

The intention of modeling concrete walls subjected to shrinkage deformation. To achieve this goal, an analytical study is carried out using the finite element method (FEM) of analysis using program ANSYS. Element (solid 65) is used to model concrete and element (link 8) to model steel reinforcement, and shrinkage phenomenon is represented by converting the measured total shrinkage strain to an equivalent temperature to calculate cracks width depending on strain generated at nodes. The overall crack pattern and the estimated crack widths are shown to have good agreement with the experimental results.

Keywords: Concrete Shrinkage, Nonlinear Analysis Shrinkage, Finite Element Analysis of Shrinkage.

أنموذج العناصر المحددة لتشققات الانكماش في الجدران الخرسانية

م.د.علي ناجي عطية
جامعة الكوفة/كلية الهندسة

ا.د.هانز جونسد
جامعة كنتاكي الامريكية

مهندس محمد حامد
جامعة الكوفة/كلية الهندسة

الخلاصة

في هذه الدراسة تم عمل أنموذج لجدار من الخرسانة المسلحة المعرض إلى تشوهات الانكماش. ولتحقيق هذا الهدف تم التحليل باستخدام العناصر المحددة ببرنامج(أنسس) . أستخدم عنصر (صلب65) ليمثل الخرسانة وعنصر (رابط 8) ليمثل حديد التسليح ، أما تمثيل ظاهرة الانكماش فتم تحويل انفعال الانكماش الكلي في جدار من الخرسانة إلى ما يكافئه من الحرارة . ولحساب عرض الشق بالاعتماد على الانفعال في العقد . وجد أن عرض الشق المتولد و المخمن له ارتباط جيد مع النتائج العملية

1. Introduction

An attempt was made by Kheder (1997) to explain the behavior and characteristics of volume change cracking of base restrained concrete walls, and adequate information was tried to enable the designer to control these cracks with the minimum possible steel reinforcement provided in cases in which the L/H (length/height) ratio of the wall is taken into consideration.

Kianoush (2007) also studied modeling of base restrained concrete wall subjected to shrinkage deformation using the finite element software (ABAQUS). Kianoush used steel reinforcement ratio for different lengths to estimate minimum crack width.

In this investigation , the ANSYS finite element computer program was used to simulate the behavior of concrete wall to shrinkage phenomenon. The finite element model uses a smeared cracking approach. Three-dimensional elements have been used to model reinforced concrete wall. This model can help to confirm the theoretical calculations as well as to provide a valuable supplement to the experimental investigations of behavior of cracks width measured by Kheder (1997).

2. Estimation of Shrinkage

The magnitude of shrinkage deformations depends on concrete mixture proportions and material properties, method of curing, ambient temperature and humidity conditions, and geometry of the concrete element. Equation of shrinkage proposed by ACI 209-08 is adopted here to estimate shrinkage strain.

3. Modeling of Concrete Shrinkage

The experimental work carried out by Kheder (1997) was used to verify the software ANSYS in numerical simulation of concrete wall shrinkage. This context, the software will be adapted to simulate the shrinkage problem and a method will be suggested to estimate the crack width, since it cannot be found directly from the program.

3.1 Experimental Work Carried out by Kheder

Kheder (1997) studied the minimum possible steel reinforcement provided in cases in which the length to height L/H ratio of the wall is taken into consideration. This wall leads to the concept of "effective reinforcement" for the control of volume change cracking in base restrained concrete wall and the details of cracks widths for the wall are shown in Fig.1 , and the properties of concrete and steel are shown in Table 1.

Kufa Journal of Engineering (K.J.E)
ISSN 2207-5528
Vol. 5, Issue 1, Dec., 2013,P.P.127-140
Printed in Iraq

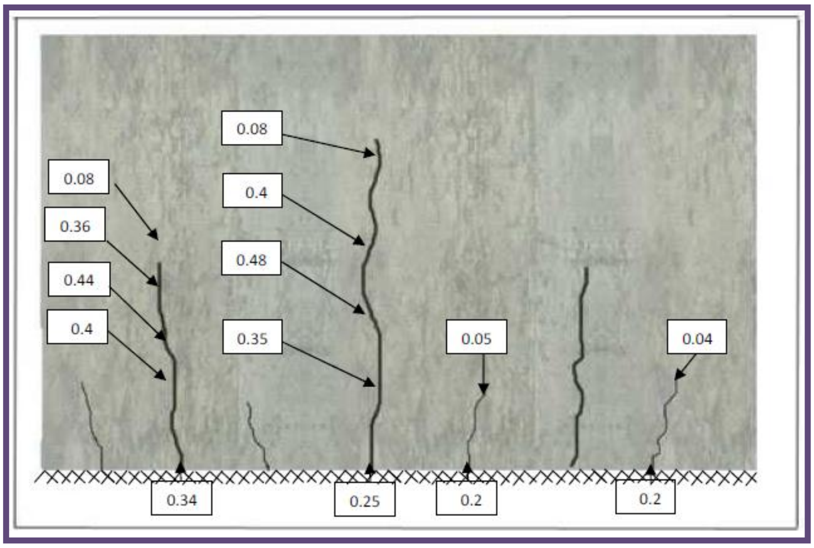

Fig.1Typical Cracks wWidths (mm) for Wall (Kheder 1997).

Table 1 Properties of Concrete and Steel (Kheder 1997).

concrete	$\acute{f_c}$, *MPa*	26
	Total free shrinkage, $\varepsilon\mu$, ε	1050
Steel	Yield Stress of Steel, f_y, *MPa*	400
	Steel ratio, ρ%	0.2
	Diameter of bars in *mm*	10

3.2 ANSYS Finite Element Model

ANSYS is well known finite element software using linear and nonlinear methods to analyze various engineering problems. To create the finite element model for any certain case using ANSYS, there are multiple tasks that have to be completed before having proper finite element model.

3.2.1 Element Types

The ANSYS software is a general multi-purpose finite element program, so there are a lot of elements used in it. The Solid65 element was chosen to model concrete, which is shown in Fig.2. This element has eight nodes with three degrees of freedom at each node – translations in the nodal x, y, and z directions. This element is capable of plastic deformation, cracking in three orthogonal directions, and crushing. The element Link8 was chosen to model steel reinforcement, which is shown in Fig.3. This element is a 3D spar element and it has two nodes with three degrees of freedom – translations in the nodal x, y, and z directions. This element is also capable of plastic deformation.

Kufa Journal of Engineering (K.J.E)
ISSN 2207-5528
Vol. 5, Issue 1, Dec., 2013,P.P.127-140
Printed in Iraq

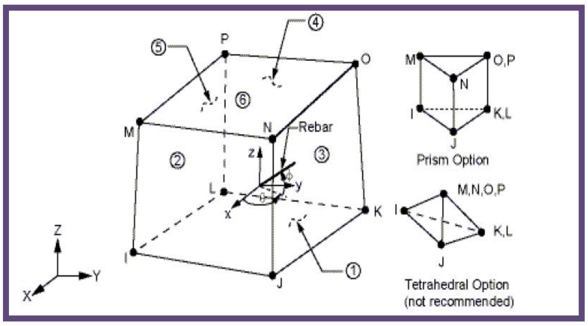

Fig.2: Solid 65 Element.

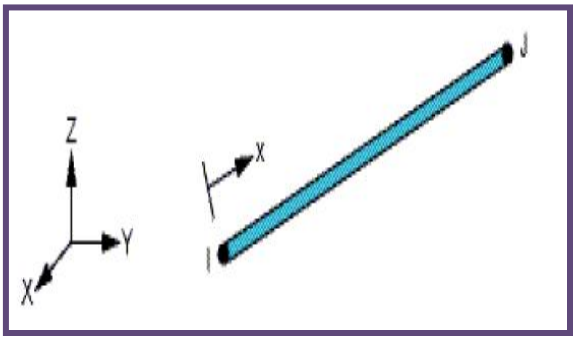

Fig.3: Link 8 Element.

3.2.2 Real Constants

The real constants for this model are shown in Table 2. Note that individual elements contain different real constants.

Table 2 Real Constants For Calibration Model.

Real constant set	**Element type**	**Constants**			
1 Concrete	Solid65		Real constant for Rebar1	Real constant for Rebar 2	Real constant for Rebar 3
		Material number	0	0	0
		Volume ratio	0	0	0
		Orientation angle θ	0	0	0
		Orientation angle β	0	0	0
2	Link8	Cross section Area(mm^2)	78.5		
Steel reinforcement		initial strain (mm/mm)	0		

Kufa Journal of Engineering (K.J.E)
ISSN 2207-5528
Vol. 5, Issue 1, Dec., 2013,P.P.127-140
Printed in Iraq

Real Constant Set 1 is used for the Solid65 element. It requires real constants for rebar assuming a smeared model. Values can be entered for: Material Number, Volume Ratio, and Orientation Angles. The material number refers to the type of material for the reinforcement. The volume ratio refers to the ratio of steel to concrete in the element. The orientation angles refer to the orientation of the reinforcement in the smeared model. ANSYS allow the user to enter three rebar materials in the concrete. The material corresponds to x, y, and z directions in the element Fig.2. The reinforcement has uniaxial stiffness and the directional orientation is defined by the user. In the present study the discrete reinforcement model is adopted. Therefore, a value of zero was entered for all real constants.

Real Constant Sets 2 is defined for the Link8 element. Values for cross-sectional area and initial strain were entered. A value of zero was entered for the initial strain because there is no initial stress in the steel.

3.2.3 Material Properties

Parameters needed to define the material models are multiple parts of the material model for each element, and can be found in Table 3.

Material Model Number 1 refers to the Solid65 element. The Solid65 element requires linear isotropic and multilinear isotropic material properties to properly model concrete. EX is the modulus of elasticity of the concrete Ec (MPa) , and PRXY is the Poisson's ratio (ν). The modulus of elasticity was based on the Eq.(4.1) (ACI 318-08 - 8.5),

$$E_c = 4700\sqrt{f'_c} \quad \text{...........} eq.(1)$$

The compressive uniaxial stress-strain relationship for the concrete model was obtained using the following equations to compute the multilinear isotropic stress-strain curve (MacGregor 1992).

$$f = \frac{E_c \varepsilon}{1 + \left(\frac{\varepsilon}{\varepsilon_0}\right)^2} \quad \text{...........} eq.(2)$$

$$\varepsilon_o = \frac{2f'_C}{E_c} \quad \text{...........} eq.(3)$$

$$E_c = \frac{f'}{\varepsilon} \quad \text{...........} eq.(4)$$

Where :-

f = stress at any strain ε ,MPa , ε = strain at stress f, and , ε_0 = strain at the ultimate compressive strength f'_c

The multilinear isotropic stress-strain relationship implemented requires the first point of the curve to be defined by the user. It must satisfy Hooke's Law.

$$E = \frac{\sigma}{\varepsilon} \quad \text{...........} eq.(5)$$

Kufa Journal of Engineering (K.J.E)
ISSN 2207-5528
Vol. 5, Issue 1, Dec., 2013,P.P.127-140
Printed in Iraq

The multilinear curve is used to help to get convergence of the nonlinear solution algorithm.

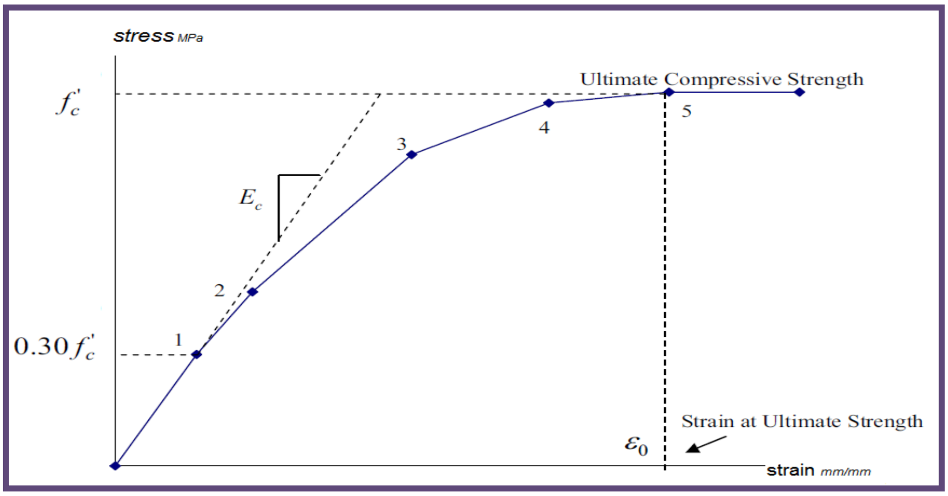

Fig. 4 Uniaxial Stress-Strain Curve.

Fig.4 shows the stress-strain relationship used in the present study and is based on work done by (Kachlakev, 2004). Point 1, defined as $(0.30f_c')$, is calculated in the linear range using Eq.4. Points 2, 3, and 4 are calculated from Eq.2 with ε_0 obtained from Eq.3. Strains were selected and the stress was calculated for each strain. Point 5 is defined at $(f_c')$and $\varepsilon_0 = 0.003$mm/mm indicating traditional crushing strain for unconfined concrete.

Table 3 Material Models for Typical Model.

Material Model Number	Element type	Material properties		
		Linear isotropic		
		EX	24019 MPa	
		PRXY	0.2	
		Multilinear Isotropic		
			Stress	Strain
		Point 1	0.000325	7.8
		Point 2	0.0007865	16.6607
		Point 3	0.001248	22.4606
		Point 4	0.0017087	25.275
		Point5	0.0021698	26
		Concrete		
		ShrCf-Op	0.4	
		ShrCf-Cl	1	
		UnTensSt	3.161	
		UnCompSt	-1	
2 Steel reinforcement	Link8	Linear Isotropic		
		EX	200,000 MPa	
		PRXY	0.3	
		Bilinear Isotropic		
		Yield Stress	400 MPa	

Kufa Journal of Engineering (K.J.E)
ISSN 2207-5528
Vol. 5, Issue 1, Dec., 2013,P.P.127-140
Printed in Iraq

Material Model Number 2 refers to the Link8 element. The Link8 element is being used for all the steel reinforcement in the wall and it is assumed to be bilinear isotropic. The bilinear isotropic material is also based on the Von Mises failure criteria. The bilinear model requires the yield stress (f_y), as well as the hardening modulus of the steel to be defined. The yield stress was defined as 400 MPa.

3.2.4 Case study and Modeling

The case study chosen in the present research is wall (1E1) of 0.15m thickness and 4 m length and 2 m height. This wall (1E1) was tested by (Kheder,1997) . Modeling this case using ANSYS will be done by creating a three dimensional solid (volume) shown in Fig.5.

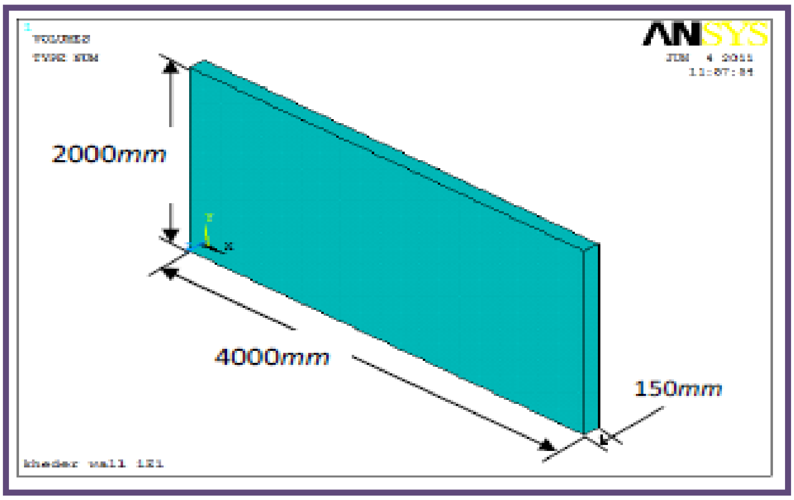

Fig.5 Volume Model.

3.2.5 Meshing

As an initial step, a finite element analysis requires creating a mesh of the model. In other words, the model is divided into a number of small elements, and after loading, stress and strain are calculated at integration points of these small elements (Bathe, 1996). An important step in finite element modeling is the selection of the mesh density. A convergence of results is obtained when an adequate number of elements is used in a model. This is practically achieved when an increase in the mesh density has a negligible effect on the results (Adams and Askenazi, 1998). Therefore, test different mesh sizes to determine an appropriate mesh density. The number of elements was increased several times until the curves in Fig. 6 flattened the horizontal.

Kufa Journal of Engineering (K.J.E)
ISSN 2207-5528
Vol. 5, Issue 1, Dec., 2013,P.P.127-140
Printed in Iraq

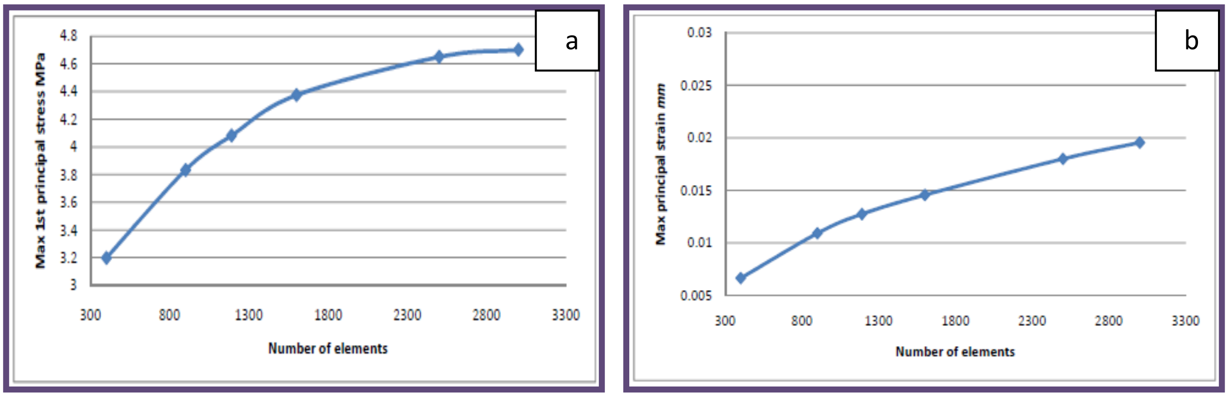

Fig.6 (a) Maximum Principal Stresses , (b) Strains of Concrete wall.

When the number of elements became larger than 2400, the results change very little, the maximum deviation of the principal stress or principal strain within ±0.5%(H. J. chen 2004) . So the use of 2500 elements for mashing the model are shown in Fig.7.

No mesh of the reinforcement is needed because individual elements were created in the modeling through the nodes created by the mesh of the model are shown in Fig.8.

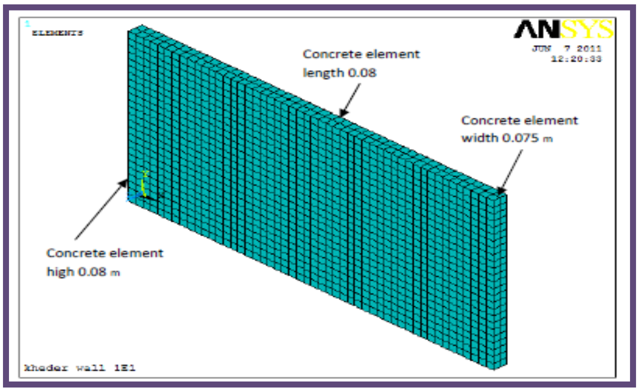

Fig.7 Mesh of Concrete.

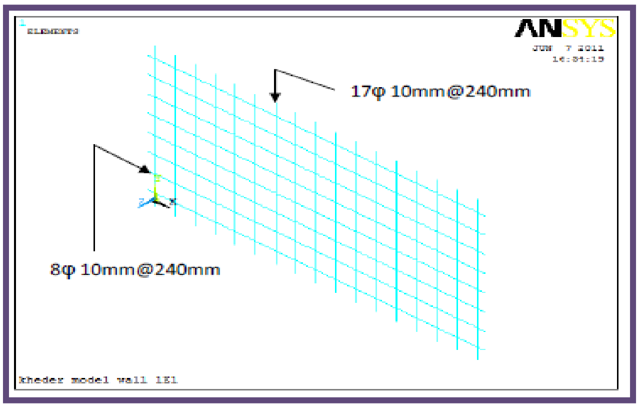

Fig.8 Modeling of Steel Reinforcement.

Kufa Journal of Engineering (K.J.E)
ISSN 2207-5528
Vol. 5, Issue 1, Dec., 2013,P.P.127-140
Printed in Iraq

3.2.6 Loads and Boundary Conditions

The concrete shrinkage as a phenomenon is usually considered as structure subjected to initial strain. Such type of loading is not available in the ANSYS, so an approach is needed to simulate the effect of shrinkage in this program. Kianoush (2007) and MABao-guo (2008) converted the measured total shrinkage strain to an equivalent temperature change. The following calculations illustrate the conversion of shrinkage strains to an equivalent temperature change:

$$T = \frac{\varepsilon_{f\,ree}}{\alpha}. = \frac{1050\times10^{-6}}{10\times10^{-6}}. = 105C^{O} \ldots\ldots\ldots\ldots\ldots\ldots\ldots\ldots eq.(6)$$

Where:

T = the equivalent shrinkage. , $\varepsilon_{\text{free}} = 1050\mu\varepsilon$(the total recorded free shrinkage). , $\alpha = 10\times10^{-6}/°C$ (the coefficient of thermal expansion) .

And according to this approach, the equivalent shrinkage temperature (T) is applied at each node in the finite element model .

Displacement boundary conditions are needed to constrain the model to get a unique solution. To ensure that the model acts the same way as the experimental wall, the ANSYS model the all DOF (degrees of freedom) are bounded (fixed) in the base of the wall as shown in Fig.9.

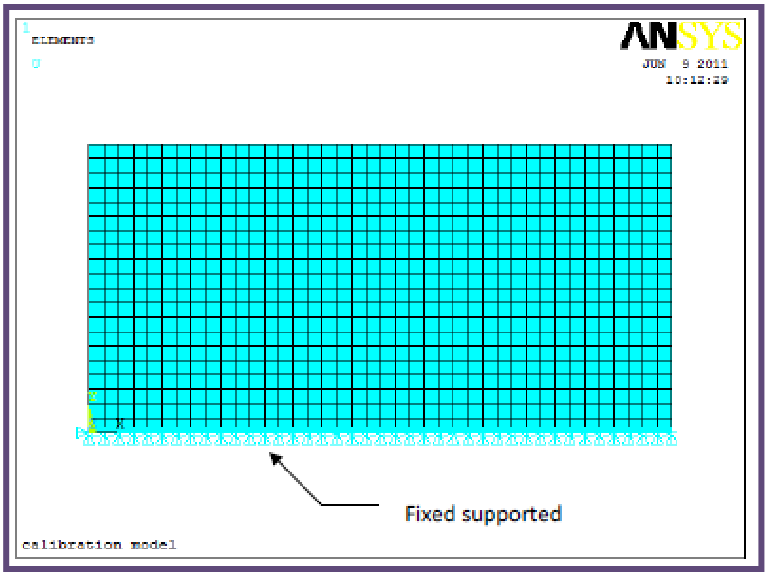

Fig.9 Boundary Condition for Supported.

3.3 Analysis Type

The finite element model for this analysis is a wall under shrinkage strain. For the purpose of this model, the static analysis type is utilized.
Typical commands utilized in a nonlinear static analysis are shown in Table 4.

Kufa Journal of Engineering (K.J.E)
ISSN 2207-5528
Vol. 5, Issue 1, Dec., 2013,P.P.127-140
Printed in Iraq

Table 4 Commands Used to Control Nonlinear Analysis.

Analysis Options	Small Displacement
Calculate Prestress Effects	No
Time at End of Loadstep	105
Automatic Time Stepping	On
Number of Substeps	105
Max no. of Substeps	10500
Min no. of Substeps	50
Write Items to Results File	All Solution Items
Frequency	Write Every Substep

All these values are set to ANSYS defaults. The commands used for the nonlinear algorithm and convergence criteria are shown in Table 5. All values for the nonlinear algorithm are set to defaults.

Table 5 Nonlinear Algorithm and Convergence Criteria Parameters.

Line Search	Off
DOF solution predictor	Prog Chosen
Maximum number of iteration	100
Cutback Control	Cutback according to predicted number of iter.
Equiv. Plastic Strain	0.15
Explicit Creep ratio	0.1
Implicit Creep ratio	0
Incremental displacement	10000000
Points per cycle	13
Set Convergence Criteria	
Label	U
Ref. Value	Calculated
Tolerance	0.05
Norm	Infinite
Min. Ref.	not applicable

The values for the convergence criteria are set to defaults except for the tolerances. The tolerances displacement is set as0.0 5 times the default values.

4. Crack Width Calculation

The computer program ANSYS is used usually to analyze structures such as walls and show the distributions of stresses and strains. The cracking pattern can be found also at each loading step, but unfortunately the crack width is not calculated. Since the crack width is the most important parameter in the control of shrinkage cracking. Concrete properties are weakness to strain tension so that any load steps lead to strain in concrete tension or compression, for tension leads to cracks this strain in node for concrete element

Kufa Journal of Engineering (K.J.E)
ISSN 2207-5528
Vol. 5, Issue 1, Dec., 2013,P.P.127-140
Printed in Iraq

is same crack width generation form shrinkage strain are shown in Fig.10 and can estimation by Eq. 7.

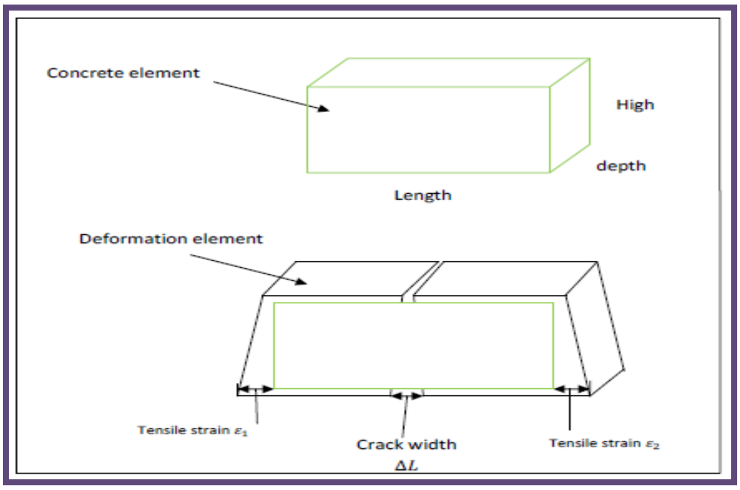

Fig.10:Method Calculated of Crack Width.

$$\omega_e = (\varepsilon_1 + \varepsilon_2) * l_e \quad \text{...} eq.(7)$$

Where :-
ω_e = crack width in concrete element. , $(\varepsilon_1 \& \varepsilon_2)$ = tensile strain in concrete element. , l_e = element length .

5. Evaluation of Cracked Model

The goal of the comparison of the finite element model and the wall tested by Kheder (1997) is to ensure that the elements, material properties, real constants and convergence criteria are adequate to model the response of the wall. Fig.12 shows the crack width calculated from ANSYS, and Fig.11 shows the location and number of cracks. Table 6 shows the deviation ratio of the finite element results of cracks width and the experimental results of Kheder (1997) wall model.

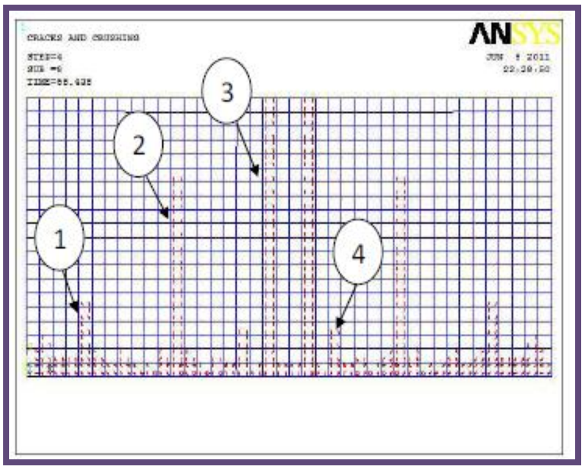

Fig.11: Location and Numbers Cracks.

Kufa Journal of Engineering (K.J.E)
ISSN 2207-5528
Vol. 5, Issue 1, Dec., 2013,P.P.127-140
Printed in Iraq

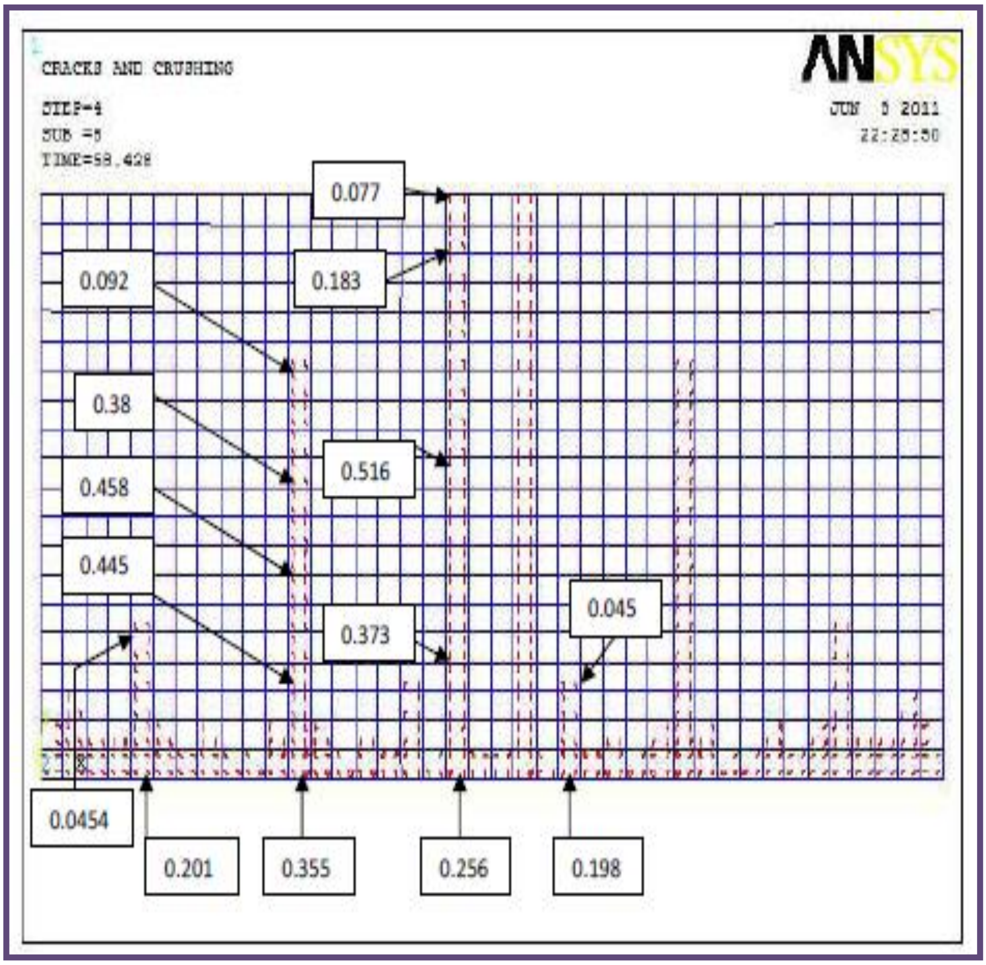

Fig.12 Crack Width for ANSYS.

Table 6 Deviation Ratio Between the Result ANSYS (FEM) and Experimental (Kheder1997).

No .of line crack	ANSYS crack width(mm)	Experimental crack width(mm)	Deviation ratio
1	0.201	0.2	0.5%
	0.0454	0.04	13.5%
2	0.355	0.34	4.41%
	0.445	0.4	11.25%
	0.458	0.44	4%
	0.382	0.36	6.11%
	0.094	0.08	17.5%
3	0.256	0.25	2.4%
	0.373	0.35	6.5%
	0.516	0.48	7.5%
	0.183	0.16	14.37%
	0.077	0.08	3.75%
4	0.198	0.2	1%
	0.045	0.05	10%

6. Conclusion

Model of shrinkage by used finite element software(ANSYS).

1. Converted the measured total shrinkage strain to an equivalent temperature.
2. Strain in node for concrete element is same crack width generation form shrinkage.

References

ACI Committee 209, "Guide for Modeling and Calculating Shrinkage and Creep in Hardened Concrete," ACI 209.2R-08, American Concrete Institute, 2008.

ACI Committee 350, "Code requirements for environmental engineering concrete structure, " (ACI 350M-06) and commentary (ACI 350RM-06).

Bathe, K. J.,"Finite Element Procedures, Prentice-Hall, Inc., Upper Saddle River, New Jersey, " 1996.

How-Ji Chen, Hsien-Sheng Peng and Yi-Feng Chen., "Numerical Analysis of Shrinkage Stress in a Mass Concrete," Journal of the Chinese Institute of Engineers, Vol. 27, No. 3, 2004, pp. 357-365.

Kachlakev D. I.,"Finite Element Analysis and Model Validation of Shear Deficient Reinforced Concrete Beams Strengthened with GFRP Laminates," California Polytechnic State University, (Cited by Anthony 2004).

Kheder GH. "A new look at the control of volume change cracking of base restrained concrete walls,"ACI Structural Journal 1997;94(3):262–71. Title no. 94-S24

Kianoush,A. , M. Acarcan,B., A. Ziari,A., "Behavior of base restrained reinforced concrete walls under volumetric change," Scince Direct Engineering Structures 30 ,2008, 1526–1534

MA Bao-guo, GAO Ying-li ., " Finite element analysis for shrinkage in the interface of functionally graded concrete segment used shield tunneling," Laboratory forsilicate materials scienece and engineering of ministry of education, Wuhan university of technology, wuhan, china ,430070, 3008/5/1 :7p.