Numerical Prediction of Bond-Slip Behavior in Simple Pull-Out Concrete Specimens

Dr.Ala'a Hussein Alwan Al-Zuhairi
Lecturer, University of Baghdad
College of Engineering-Civil Department
Email: alaalwn@yahoo.com

Wjdan Dhaif Sahi Al-Fatlawi
Ass. Lectuer, Ministry of Municipalities and Public Works-Sewerage Directorate-Design Department
Email: wjdan_civil@yahoo.com

ABSTRACT

In this study the simple pullout concrete cylinder specimen reinforced by a single steel bar was analyzed for bond-slip behavior. Three-dimension nonlinear finite element model using ANSYS program was employed to study the behavior of bond between concrete and plain steel reinforcement. The ANSYS model includes eight-noded isoperimetric brick element (SOLID65) to model the concrete cylinder while the steel reinforcing bar was modeled as a truss member (LINK8). Interface element (CONTAC52) was used in this analysis to model the bond between concrete and steel bar. Material nonlinearity due to cracking and/or crushing of concrete, and yielding of the steel reinforcing bar were taken into consideration during the analysis. The accuracy of this model is investigated by comparing the finite element numerical behavior with that predicted from experimental results of three pullout specimens. Good agreement between the finite element solution and experimental results was obtained.

Key words: bond-slip relationship, plain bar, FEM, ANSYS program, pull-out test.

التخمين العددي لتصرف علاقة الربط-الأنزلاق في نموذج سحب خرساني بسيط

د.علاء حسين علوان الزهيري
مدرس

وجدان ضايف ساهي الفتلاوي
مدرس مساعد

الخلاصة

في هذة الدراسة تم تحليل نموذج سحب خرساني اسطواني بسيط مسلح بسيخ حديد املس منفرد في مركز الاسطوانة بواسطة طريقة التحليل اللاخطي للعناصرالمحددة ثلاثية الابعاد باستعمال برنامج نظام التحليل الحاسوبي ANSYS لدراسة تصرف الربط بين الخرسانة وحديد التسليح الاملس. اذ تم استعمال عناصر طابوقية ثلاثية الأبعاد ذات ثمان عقد نوع Isoperimetric تسمى (Solid65) لتمثيل خرسانة الاسطوانة بينما تم تمثيل حديد التسليح باستعمال العنصر المحوري(Link8). اما المنطقه البينيه(منطقة التماس بين الخرسانة والحديد) فقد تم تمثيلها باستعمال عنصر التداخل (Contac52) والذي يربط العناصر الطابوقية والعناصر المحورية لتمثيل سلوك الربط بين الخرسانة وحديد التسليح مع الأخذ بنظرالاعتبار التصرف أللاخطي لكلا المادتين، والذي يشمل تشقق و/أوتهشم الخرسانة وخضوع حديد التسليح. لقد تمت مقارنة نتائج التحليل العددي مع النتائج العملية لثلاث نماذج سحب نموذجية للتحقق من دقة التمثيل اذ تم الحصول على تقارب وتوافق بدرجة جيدة بين نتائج التحليل العددي والنتائج العملية.

الكلمات الرئيسية: علاقة الربط- الأنزلاق، قضيب أملس، طريقة العناصر المحددة، برنامج ANSYS، فحص السحب

1. INTRODUCTION

The ability of using of reinforced concrete as a structural material is derived from the combination of concrete which is strong in compression with reinforcing steel that is strong and ductile in tension. Maintaining composite action requires transfer of load between the concrete and steel. This load transfer is referred to as bond. Bond stress is defined as the shear stress acting on the surface between the bar and concrete in the direction of the bar. It governs some phenomena in reinforced concrete such as cracking and tension-stiffening during the loading stage till failure. Bond can be activated under various actions like pure tension, pull-out, push-in …etc. Selection of any one of these loading methods depends on many variables such as characteristics of reinforcing bar, concrete physical properties, and member geometry.

The idealization of bond in finite element method (FEM) allows considering several phenomena: plasticity, contact, cracking …etc.). This is may be the reason why FEM has been applied to bond modeling by several researchers starting by the pioneer work of Ngo and Scordelis, 1967 to the most recent advancements (Bamonte et al, 2003, Sezen and Mohle, 2003, Jendele and Cervenka, 2006, and Khalfallh and Ouchenane, 2007).

In this paper, the finite element method is used to investigate bond behavior of pull-out cylinder reinforced with a plain steel bar. The analysis is made utilizing the computer program ANSYS 9.0.

2. THE DETAILS OF TEST SPECIMEN:

The pull-out cylinder specimen (150×300mm) shown in Fig.(1) was used in this study. The bonded length (*L*) was taken equal to 12times the bar diameter. It was assumed that the slip is constant along the bonded length (*L*) of the steel bar. Consequently, the bond stress (*u*) is uniform. Hence, the bond stress can be calculated from equilibrium condition as follows:

$$A_b \times f_s = L \times \pi \times \phi_b \times u$$

$$\pi \times \frac{\phi_b^2}{4} \times f_s = L \times \pi \times \phi_b \times u \qquad (1)$$

$$u = \frac{\phi_b}{4L} \times f_s$$

Where,

u = average bond stress.
L = bonded length.
ϕ_b = steel bar diameter.
A_b = cross section area of steel bar.
f_s = tensile stress in steel.

Since L was taken as equal to $12\phi_b$, therefore:

$$u = \frac{f_s}{48} \qquad (2)$$

Three specimens were tested experimentally by varying the diameter of the steel plain bar as (10, 12, and 16)mm. The results of the test were used in the comparison that made with the results of the finite element analysis.

3. FINITE ELEMENT MODEL:

3.1 Element Types

The solid brick element, SOLID65, was used to model the concrete in ANSYS program. The solid element has eight nodes with three degrees of freedom at each node, translations in the nodal x, y, and z directions. The element is capable of plastic deformation, and cracking in three orthogonal directions. The two-noded LINK8 bar (truss) element was used to model the steel reinforcement. At each node, the degrees of freedom are identical to those for the SOLID65. The element is also capable of plastic deformation. Point to point contact element (CONTAC 52) was used to model bond-slip of reinforcement bar in the present study. The element joins two surfaces that may maintain or break physical contact and may slide relative to each other. Also, it is capable of supporting only compression in the direction normal to the interface between the two surfaces and Coulomb shear-friction in the tangential direction. The 3-D point-to-point contact element has three degrees of freedom at each node in the element coordinate system. The orientation of the interface is defined by the node locations.

3.2 Material Properties

Concrete: SOLID65 elements are capable of predicting the nonlinear behavior of concrete materials using smeared crack approach Willam and Warnke, 1975. The smeared crack approach has been adopted widely in the last decades. Concrete is a quasi-brittle material and has very different behaviors in compression and tension. The stress-strain relation for concrete in compression was described by multilinear elastic model as shown in Fig. (2). Based on the compressive strength of concrete, the stress-strain relationship was obtained using the following equation (MacGregor, 1992)**:**

$$f = \frac{E_c \varepsilon}{1 + \left(\dfrac{\varepsilon}{\varepsilon_o} \right)^2} \qquad (3)$$

Where,

f = stress at any strain ε, MPa.

ε = strain at stress f.

ε_o = strain at the ultimate compressive strength f'_c, $\varepsilon_o = \dfrac{2 f'_c}{E_c}$.

E_c = concrete elastic modulus

$E_c = 4700\sqrt{f'_c}$ MPa (ACI 318 Code).

The failure surface of the concrete proposed by Willam and Warnke, 1975, is adopted in this study.

Steel Reinforcement: a multilinear isotropic hardening with von- Mises yield criterion model is used to define the material properties of steel bar. The tensile stress-strain response of steel based on the test data shown in Fig. (3) is used in the present analysis by picking the values in data table of ANSYS 9.0 program.

Bond- Slip Model: The interface element (CONTAC 52) is capable of supporting only compressive forces in the direction normal to the interface surface and shear (Coulomb friction) in the tangential direction. The interface element (CONTAC 52) may have one of three conditions:closed and stuck, closed and sliding, or open. The force-deflection relationships for the interface element (CONTAC 52) can be separated into normal and tangential (sliding) directions as shown in Fig.(4).

The element (CONTAC 52) is defined by two stiffnesses: normal stiffness (k_n) and tangential stiffness (k_s). The normal stiffness (k_n) is calculated from the following equation (Fardis and Buyukozturk, 1980).

$$k_n = \frac{d}{0.7}\left[\pi E_s K_f^3 \left(\frac{f'_c}{27.4} \right)^6 \right]^{1/4} \text{ (MPa/mm)} \qquad (6)$$

where:

E_s= Elastic Modulus for steel bar.

K_f= is the foundation modulus, depending on the tensile stress in the reinforcement as:

$$K_f = 820 - 1.17 f_s \quad \text{N/mm}^3 \qquad (7)$$

where, f_s is the tensile stress in the reinforcement (MPa).

The tangential (sticking) stiffness (k_s) is found by multiplying the friction by normal stiffness.

The input data for the concrete, steel reinforcement, and interface element are summarized in Table (1).

Table (1): Materials properties of the pull-out specimens

Material type	Parameter	Values for		
		Case1	Case2	Case3
Concrete	Compressive strength, f'_c (MPa)	31.0		
	Tensile strength, $f_t = 0.1 f'_c$ (MPa) (Chen, 1982)	3.1		
	Young modulus, E_c (MPa)	26187.6		
	Poisson’s Ratio, ν (assumed)	0.15		
Reinforcing steel	Actual diameter (mm)	10.85	12.60	15.75
	Yield stress, f_s (MPa)	369.4	378.6	300.9
	Young modulus, E_s (MPa)	210083.1	210136.3	206859.0
	Poisson’s Ratio, ν (assumed)	0.3		
	Bonded length = $12d_b$ (mm)	130	150	190
Interface (contact)	Coefficient of Friction μ (assumed)	0.3		

3.3 Finite Element Modeling

The specimen is a concrete cylinder of 150mm diameter, and 300mm length, with single concentric plain bar. Contact elements (interface elements) are alternatively used at the interface between the concrete and the steel bar. To obtain good results from the concrete element (Solid 65) is arranged in a rectangular mesh i.e., the mesh is set-up such that square or rectangular elements are created. The meshing of reinforcing bar has corresponded to the meshing of concrete volume. The boundary conditions for the geometric model are applied by fixing the nodes at the top surface of cylinder in three directions except the nodes adjacent to the steel bar and the nodes of unbounded length fixed in two directions (x and y). The finite element model of the cylinder specimen is shown in Figs. (5) and (6).

4. RESULTS AND DISCUSSION:

Bond-slip relations have been established from the finite element analysis and compared with the experimental results as shown in Fig. (7) for three cases of verification. All curves have mostly the same trend.

The bond-slip curves are obtained for plain bars of 10mm, 12mm and16mm diameter during the loading stage only. No slip greater than that shown in these curves could be obtained because after this stage the bar is pulled continuously out of the concrete cylinder. These figures clarify that bond stress is composed of two components. At initial stages of loading the main parts of the bond are generated from chemical adhesion between the concrete and steel reinforcement. Typical values of bond stress ranging from (0.03 to 0.9) MPa. The generation of this component of bond stress is not accompanied by a significant slip between the reinforcing steel and the surrounding concrete. As the applied tensile force increases, the second component of the bond will start due to friction developement. The role of the chemical bond is more pronounced in the smooth bars and its effect decreases or diminishes as the reinforcing bar diameter increases.

Table(2) shows experimental and nummerically calculated loads that measured and predicted at bond failure. It is observed that the mimum force required overcoming the bond strength mobilized between reinforcing bar and surrounding concrete is increased with increasing of bar diameter. However, this trend is obosite when the tensile stress in bar material (steel) is considered. between results predicted from finite element analysis and those obtained experimentally may be attributed to sophisticated method in determining of the normal stiffness of the interfce (contact) element. Fig.(8) shows the average bond-slip relationships for different diameters of steel bars embeded in concrete cylinder specimens of the compressive strength. It is cleared that the bond stress decreases with increasing of bar diameter

Table (2): Experimental and predicted bond failure loads for three cases

Test case	Nominal size of steel bar (mm)	Actual diameter of steel bar (mm)	Expermental results			F.E. analysis results			Error, E (%) $E = \frac{F.E - Exper}{Exper} \times 10$
			Max. force (kN)	Max. stress in steel (MPa)	Average bond stress (MPa)	Max. force (kN)	Max. stress in steel (MPa)	Average bond stress (MPa)	
1	10	10.85	16.156	174.8	3.64	17.000	183.9	3.83	5.2
2	12	12.60	18.722	150.2	3.13	20.500	164.5	3.43	9.4
3	16	15.75	26.430	135.7	2.83	27.000	138.7	2.89	2.2

.

5. FAILURE MODE

The pull-out failure is observed in both experimental tests and finite element analysis. No cracking of the concrete is indicated in any of test specimens as shown in Fig. (9). On the other hand, no yielding was occurred in steel bar. Fig.(10) shows the pull bar and deformation of interface elements which connects the concrete and steel reinforcement.

6. CONCLUSIONS

1. The trend of bond-slip realation was found inpependent of bar diameter.
2. The use of interface (contact) element through analytical study helps the numerical solution to exhibit a good agreement with experimental results.
3. The differences (errors) between the results predicted by FEA and those obtained experimentally is attributed to the difficulty of determining the normal stiffness of the interface element
4. For both experimental and Nummerical analyses, bond strength increases by decreasing the diameter of steel bar embeded in concrete cylinder specimens of the same compressive strength.
5. Chemical adhesion decreases with the increase of embedded bar diameter. This resistance is observed at the early stage of loading when the pullout force is applied without any slippage of the reinforcing bar.
6. The pull-out failure is the predominant type of failure observed in specimens. Neither cracking of concrete nor yielding of steel bars was indicated.

REFERENCES:

ANSYS Theoretical Manual, Version 11.0.

Bamonte, P., Coreonelli, D., and Gambarova, P.G., “Smooth Anchored Bars in NSC and HPC: A Study on Size Effect”, Journal of advanced concrete technology, Vol. 1, No.1 January 2003, pp.42-53.

Chen, W. F., “Plasticity in Reinforced Concrete” McGraw-Hill Book Company, Inc., New York, pp.474, 1982.

Fardis, M., and Buyukozturk, M.,“Shear Stiffness of Concrete by Finite Elements”ASCE Proceedings, Journal of Structure Division, Vol. 106, No.St6, June 1980, PP.1311-1327.

Hong, S. and Park, S. "Uniaxial Bond Stress-Slip Relationship of Reinforcing Bars in Concrete", 2012, Advances in Materials Science and Engineering, Vol. 2012 Article ID 328570, 12 pages, Hindawi Publishing Corporation.

Jendele, L.,and Cervenka, J., **“**Finite Element Modeling of Reinforcement With Bond”, Computer and structures Journal 84 ,August 2006, pp. 1780-1791.

Khalfallah, S., and Ouchenane, M., “Prediction of Bond Between Steel and Concrete by Numerical Analysis”, The Open Civil Engineering Journal, University of Jijel, Algeria, 2008, Vol.2, pp. 1-8.

Khalfallah, S., and Ouchenane, M., **“**A Numerical Simulation of Bond for Pull-out Tests the Direct Problem”, Asian Journal of Civil Engineering (Building and Housing), Vol.8, No.5, 2007, pp. 491-505.

MacGregor, J.G., "Reinforced Concrete Mechanics and Design", Prentice-Hall, Inc.,Englewood Cliffs, NJ, 1992

Ngo, D. and Scordelis, A. C., "Finite Element Analysis of Reinforced Concrete Beams**"**, ACI Journal, Vol. 64, No. 3, March1967, pp. 152- 163.

Sezen, H., and Moehle, J.P., **"**Bond-Slip Behavior of Reinforced Concrete Members", The Earth Engineering Research Centers Program of the National Science Foundation at the University of California, Berkeley, 2003.(Internet)

Willam,K., and Warnke, E., **"**Constitutive Model for the Triaxial Behavior of Concrete**"**, Proceedings, International Association for Bridge and Structural Engineering, Vol.19, ISMES, pp.174, Bergamo, Italy, 1975.

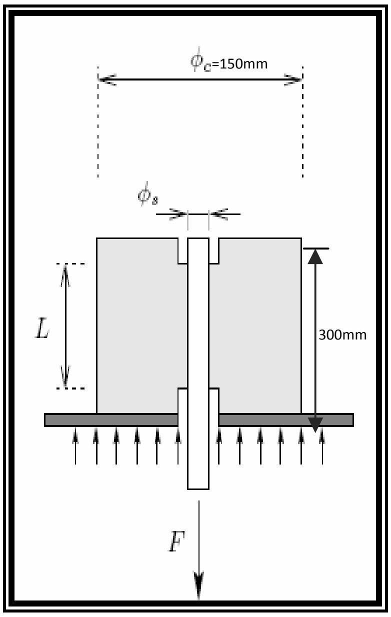
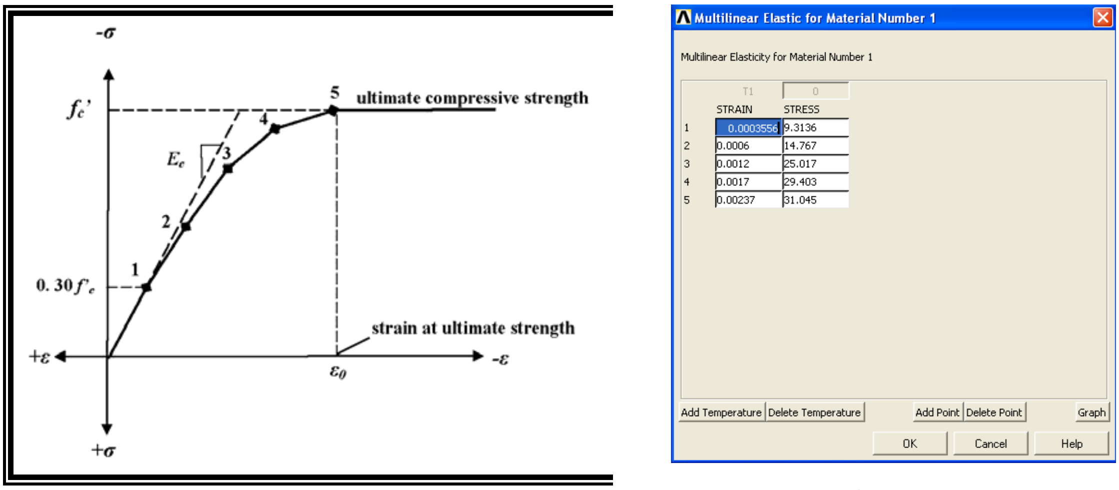


Fig. (1): Pull-out specimen



Multilinear Elasticity for Material Number 1

T1 0

	STRAIN	STRESS
1	0.0003556	9.3136
2	0.0006	14.767
3	0.0012	25.017
4	0.0017	29.403
5	0.00237	31.045

(a): Stress-Strain curve of concrete

(b): Data table for stress-strain curve of concrete

Fig. (2): The adopted stress-strain curve of concrete in ANSYS 9.0 program

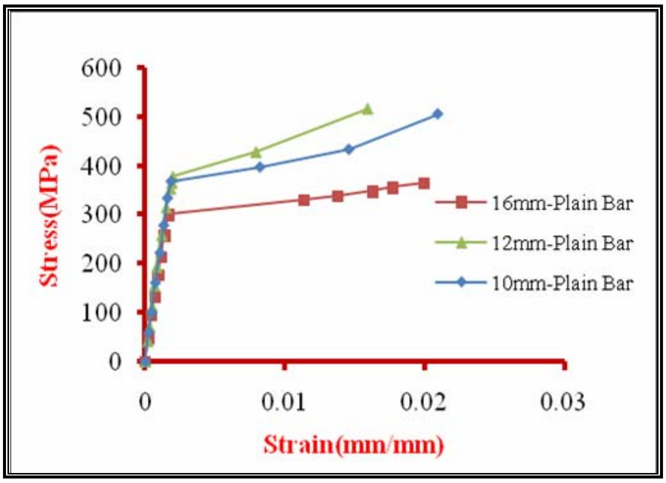


Fig. (3): Stress-strain diagrams for tested reinforcing steel bars

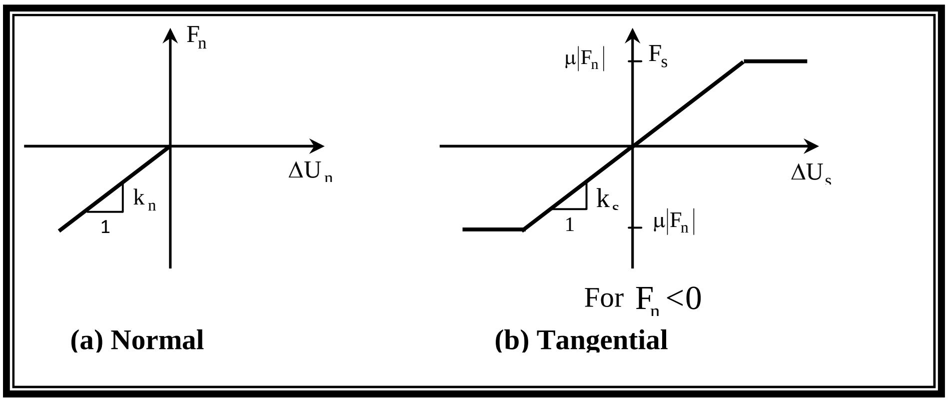
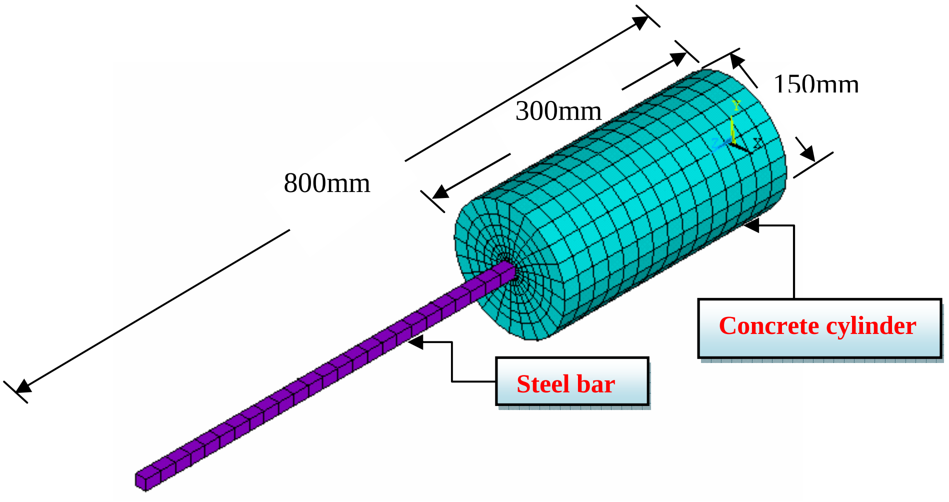
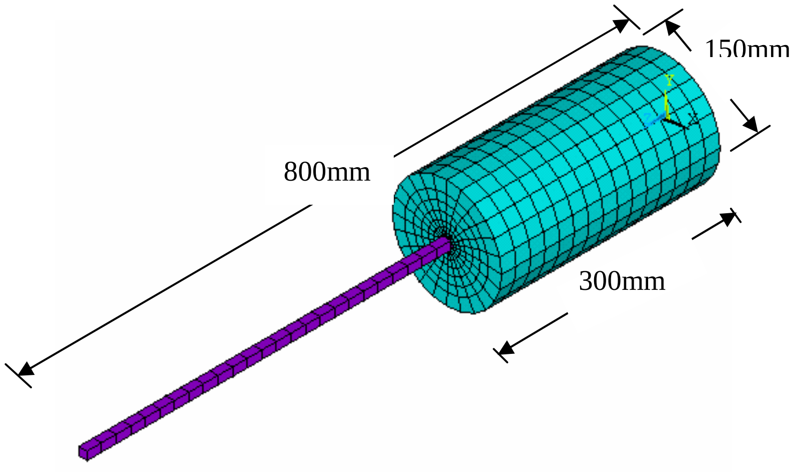


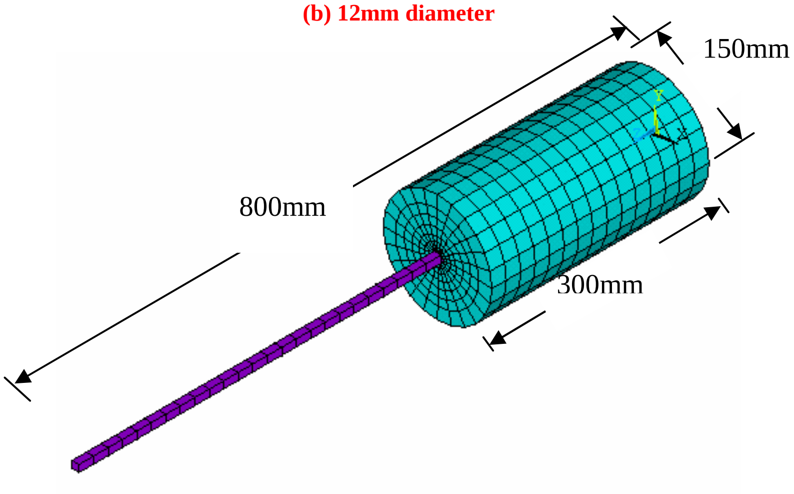
Fig. (4): Interface force-deflection relationship ANSYS



(a) 16mm diameter



(b) 12mm diameter



(c) 10mm diameter

Fig. (5): Finite element mesh of pull-out specimens created using ANSYS 9.0 program

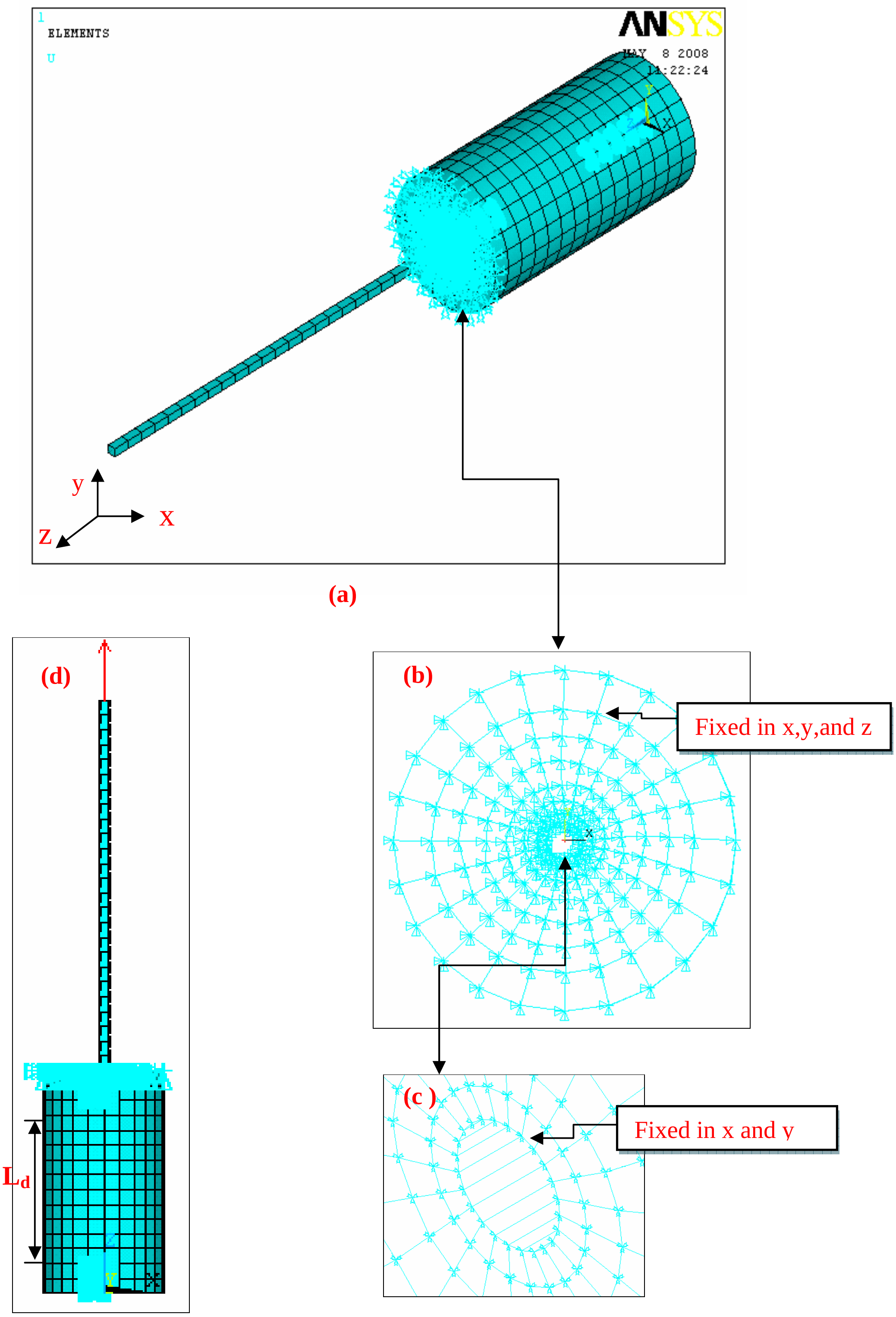


Fig.(6) :Applying boundary condition and force
(a, b, and c) Applying boundary conditions
(d) Applying development length and force

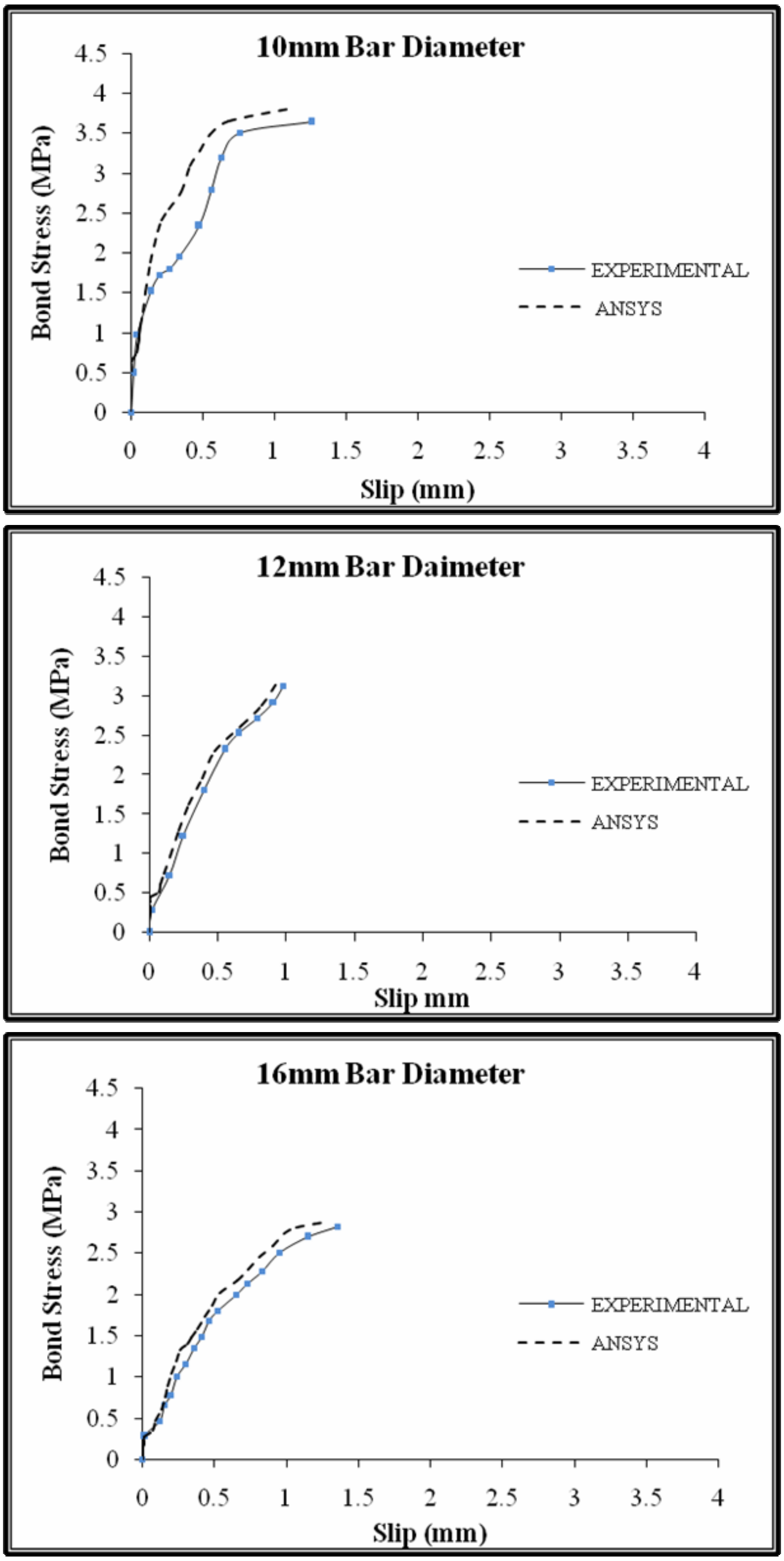


Fig. (7): Bond-slip relationships of concrete compressive strength 31.045MPa and for different steel bar diameters

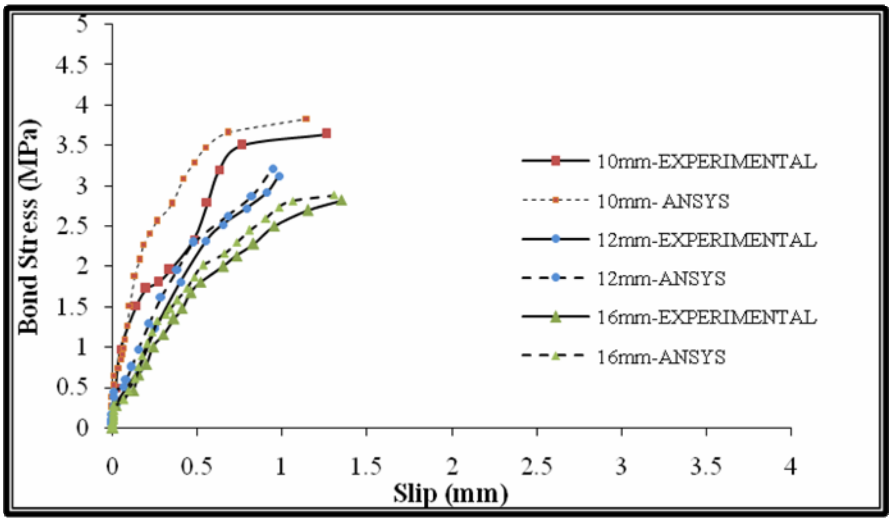


Fig. (8): Bond stress-slip relationship for compressive strength 31.045MPa with different diameters

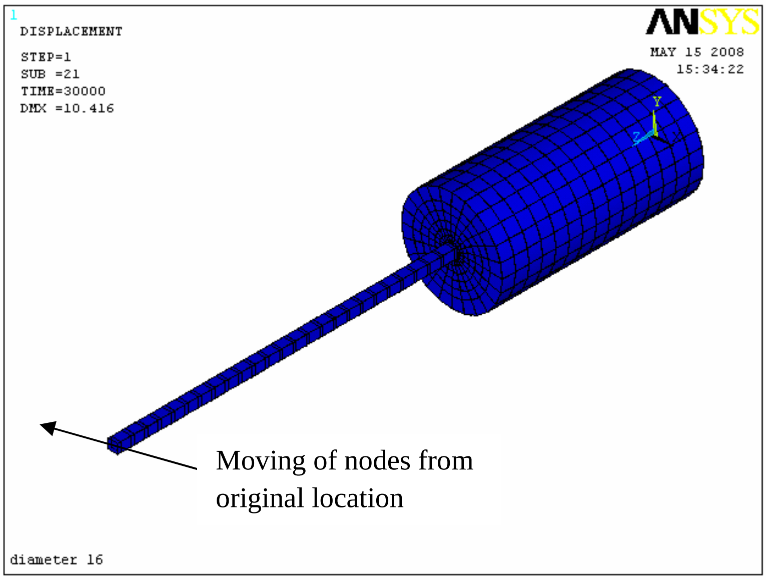
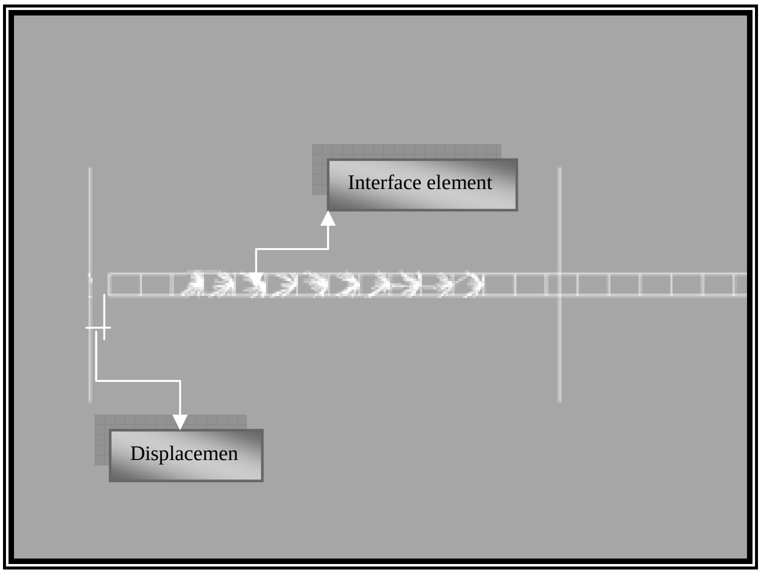
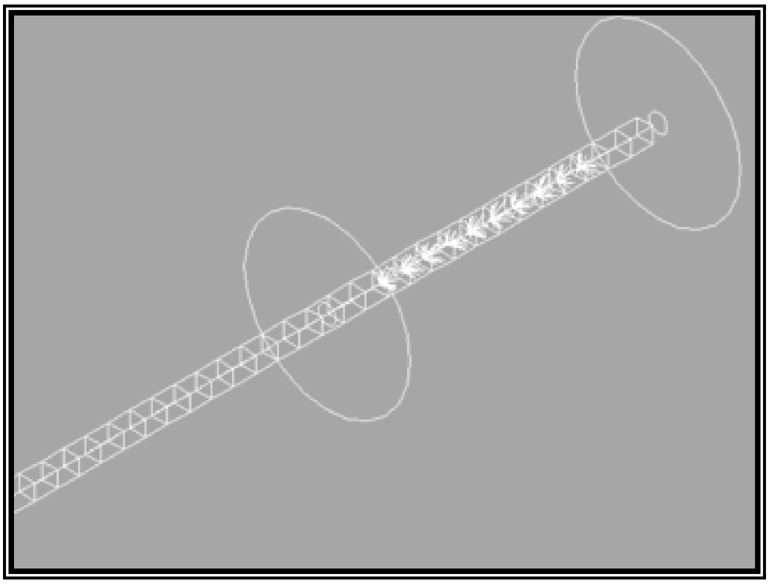


Fig. (9): The deformed and un-deformed shape of the model.



Side view (a)



(b) Iso view

Fig. (10): Pull steel bar and deformation of interface elements.