International Journal of Engineering Science Invention
ISSN (Online): 2319 – 6734, ISSN (Print): 2319 – 6726
www.ijesi.org ||Volume 5 Issue 4|| April 2016 || PP.49-54

Finite Element Analysis of Effect of Punching Shear in Flat Slab Using Ansys 16.0

Yogesh A.Chaudhari[1],Prof.G.B.Katti[2]
[1]PG Student, Department of Civil Engineering,Sapkal Knlowedge Hub College of Engineering,Nashik.
[2]Prof.G.B.Katti, Department of Civil Engineering,Sapkal Knlowedge Hub College of Engineering,Nashik.

ABSTRACT : *Finite element analysis is useful numerical technique to solve various structural problems. In this paper FEA model of slab column connection is model using ANSYS 16.0 . Punching shear failure is a major problem encountered in the design of reinforced concrete flat plates. The utilization of shear reinforcement via shear studs or other means has become a choice for improving the punching shear capacity .The obtained results indicate that, the proposed shear reinforcement system and drop panel has a positive effect in the enhancement of both the punching shear capacity and the strain energy of interior slab–column connection of both normal and high strength concrete. The general finite element software ANSYS can be used successfully to simulate the punching shearbehaviour of reinforced concrete flat plates.*
KEYWORDS –*ANSYS, Drop Panel, Flat Slab, Punching.*

I. Introduction

Punching shear is a critical design factor of reinforced concrete flat plates since it is associated with brittle failure. Many alternative reinforcement systems had been introduced in literature; e.g., shear studs, bent bars, in order to enhance the punching shear and the strain energy of slab–column connection. The punching shear strength and deformation capacity are strongly influenced by the type and characteristics of the shear reinforcing system. Flat slab is an ideal structural form for architects and contractors. Its flush soffit makes the formwork construction, wiring and ducting work easy. Without using beams, flat slab provides more headroom or lower storey height. It can thus allow for more storeys than other types of slab systems within the same building height. But flat slab has inherent weaknesses. The connections between the floor slab and column in a flat slab structure are generally the most critical part as far as the strength is concerned because it is a region where large moments and shear forces are concentrated.

II. System Development

ANSYS 16 is useful to finite element simulation for RCC structure ewe use Solid 186 for concrete, link8 for Rebar (Reinforcement), Conta 174 and Targe 173 to define contact between them.

2.1 SOLID186 Element Description

SOLID186 is a higher order 3-D 20-node solid element that exhibits quadratic displacement behaviour. The element is defined by 20 nodes having three degrees of freedom per node: translations in the nodal x, y, and z directions. The element supports plasticity, hyper elasticity, creep, stress stiffening, large deflection, and large strain capabilities. It also has mixed formulation capability for simulating deformations of nearly incompressible elastoplastic materials, and fully incompressible hyper elastic materials.

2.2 CONTA 174 and TARGE170

The 3-D contact surface elements (CONTA173 and CONTA174) are associated with the 3-D target segment elements (TARGE170) via a shared real constant set. ANSYS looks for contact only between surfaces with the same real constant set. For either rigid-flexible or flexible-flexible contact, one of the deformable surfaces must be represented by a contact surface.

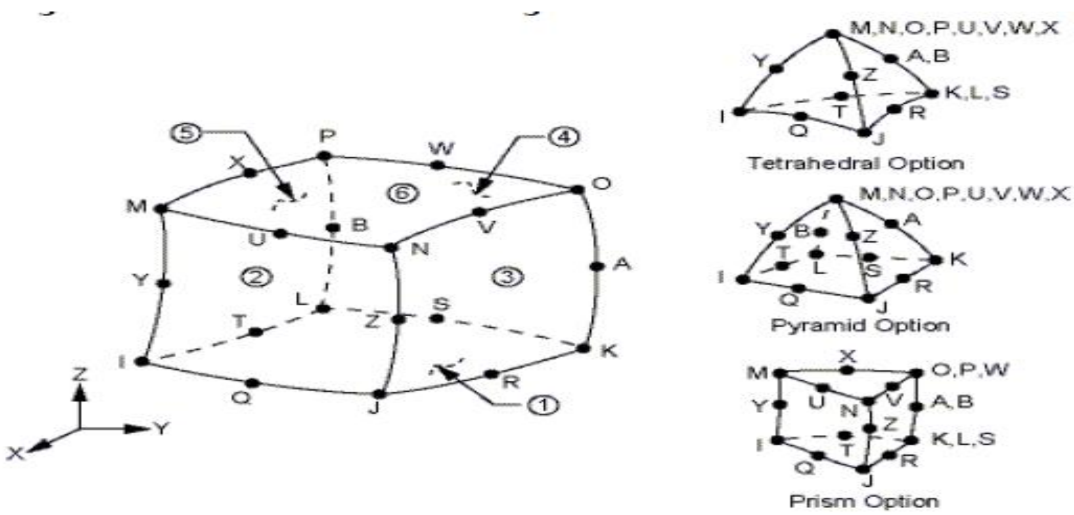


Fig. 1

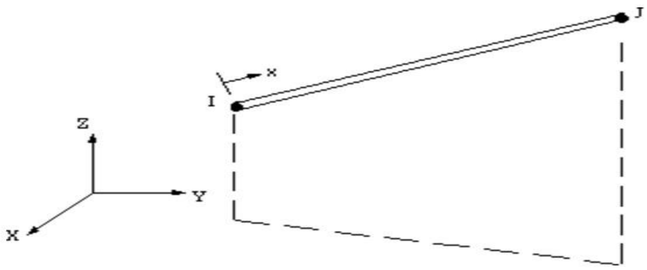


Fig 2

If more than one target surface will make contact with the same boundary of solid elements, you must define several contact elements that share the same geometry but relate to separate targets (targets which have different real constant numbers), or you must combine two target surfaces into one (targets that share the same real constant numbers).

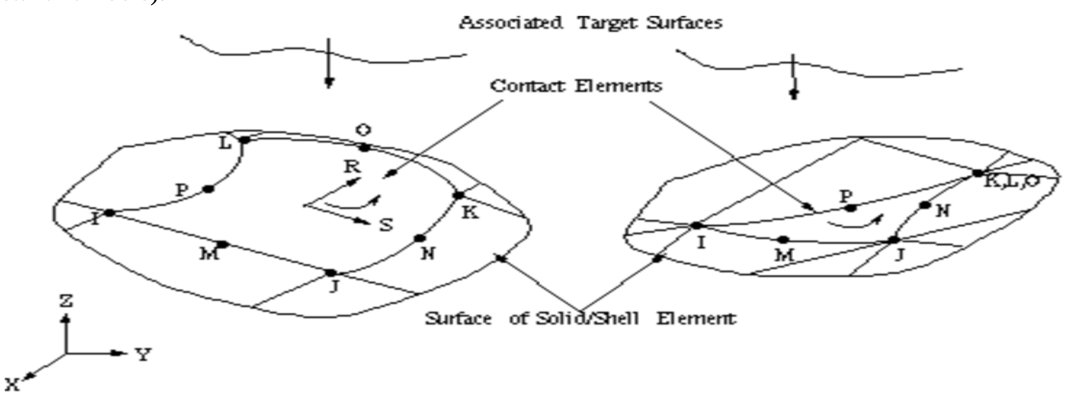


Fig 3

2.3 Finite Element meshing in ANSYS

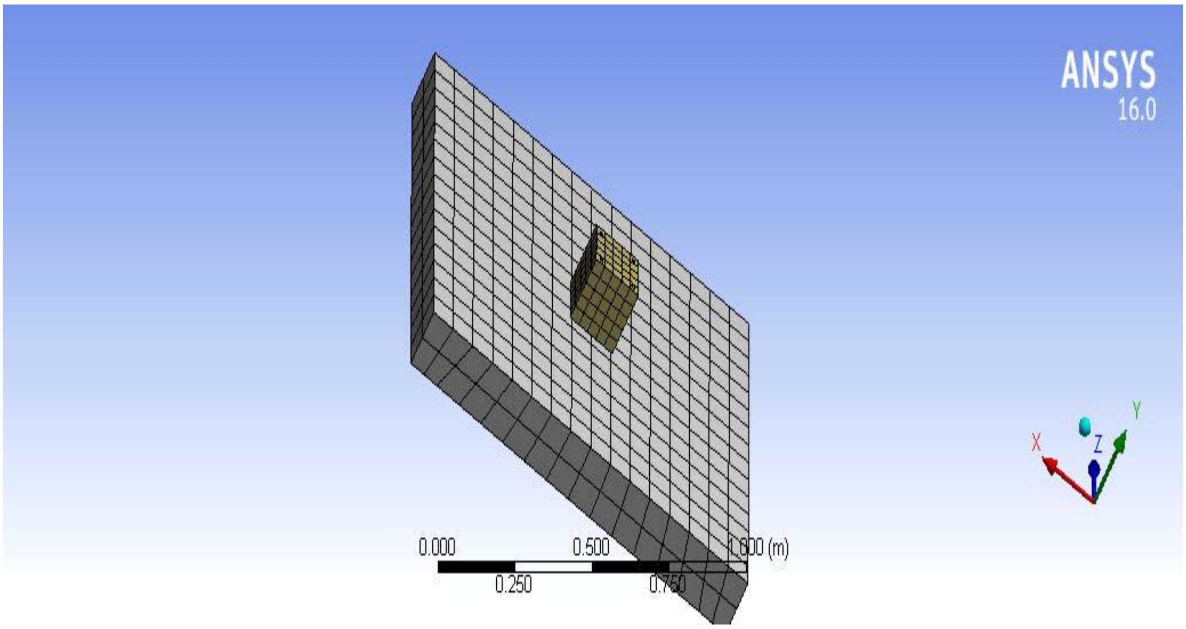


Fig4

III. Problem Statement

The Finite Element Model consisted of square flat plates 1200 mm length and 140 mm thick with 160 mm square reinforced concrete column stubs extending 160 mm above the plate. All the slabs were identical in dimensions. The reinforcement was distributed uniformly throughout the width of the slab as shown in model.

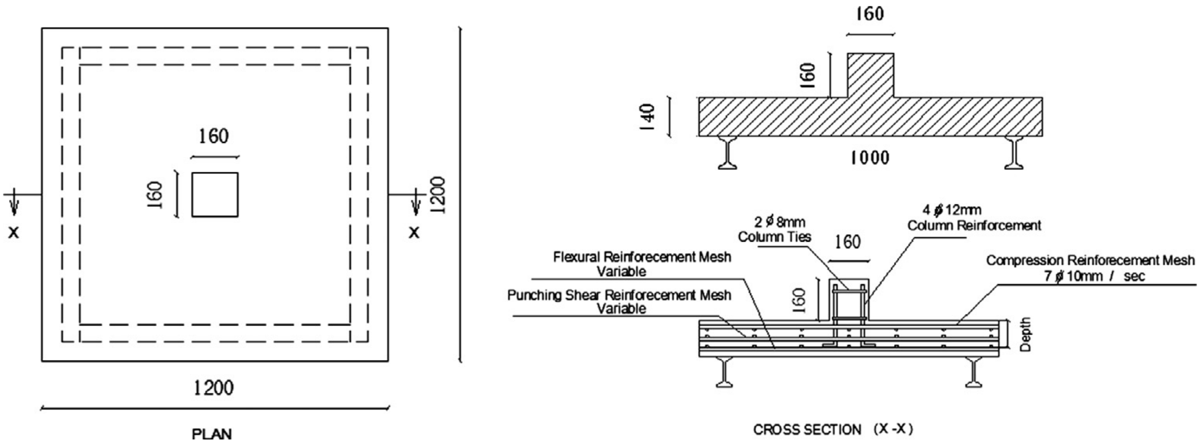


Fig 5

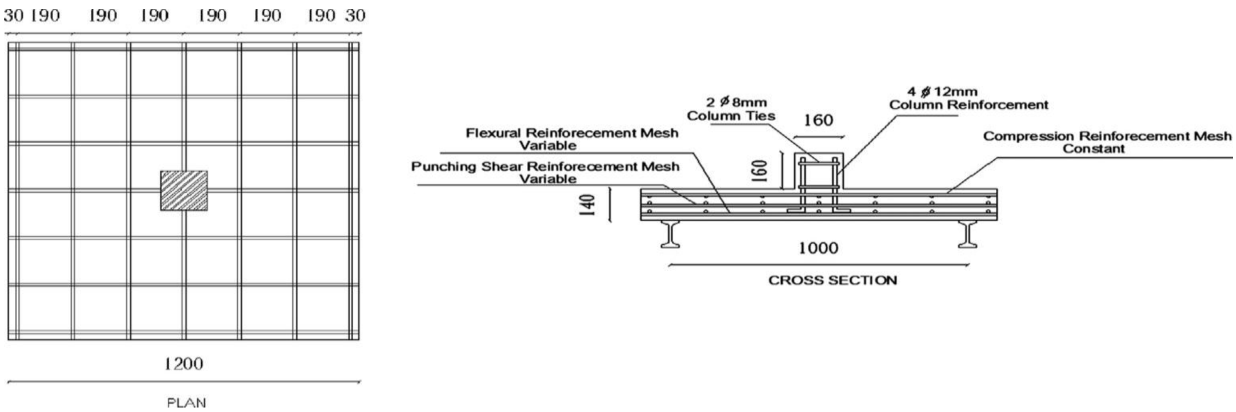
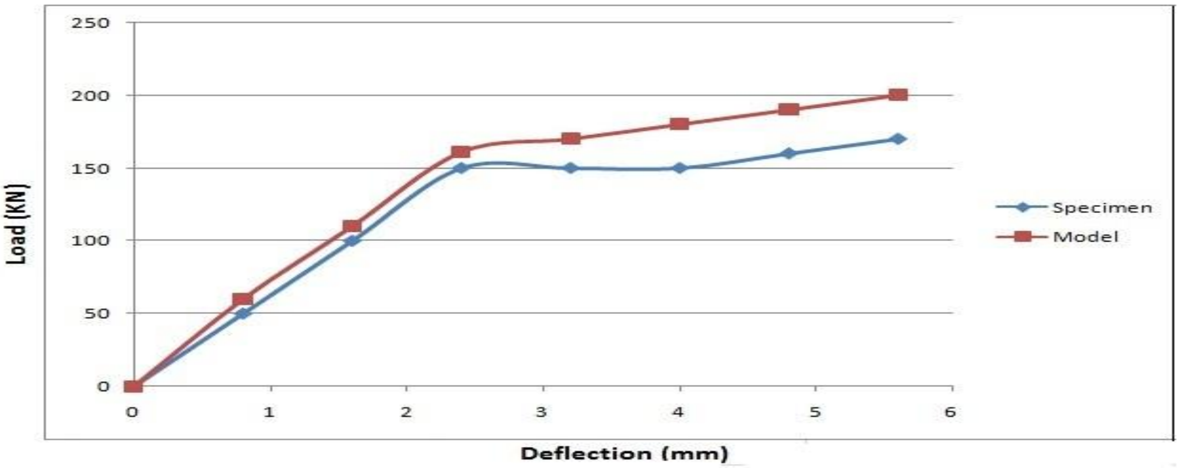


Fig 6

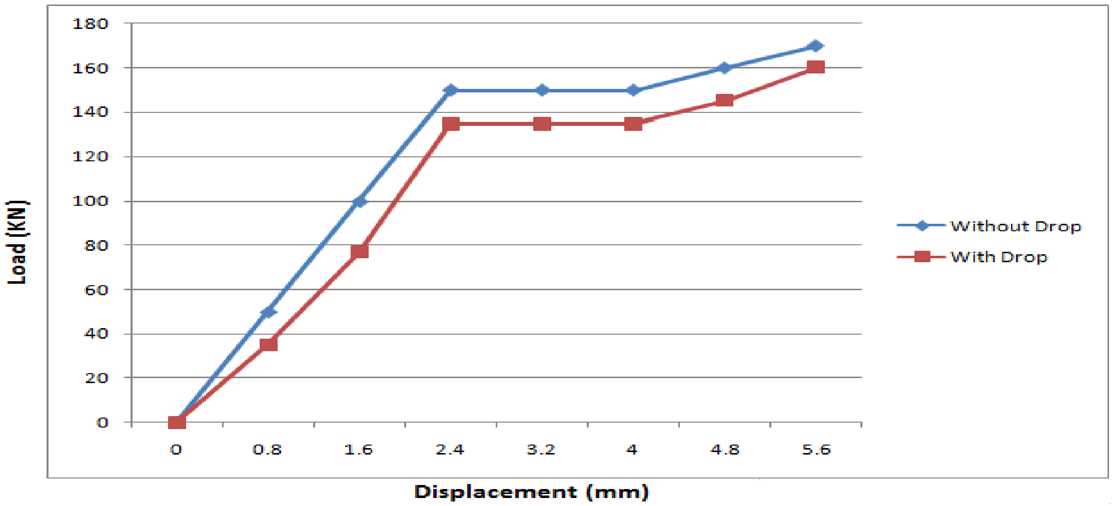
IV. Performance Analysis:

Results of the Finite Element analysis were validated with experimental test results(5).



Graph 1

The percentage error is found to be 20%. for non-linear results. For statics linear analysis analytical and experimental results are nearly same. First crack in specimen is found at 150kN.The normal stress, shear stress, shear strain, total deformation, and strain energy results for 150kN are shown in fig9,fig10.



Graph 2

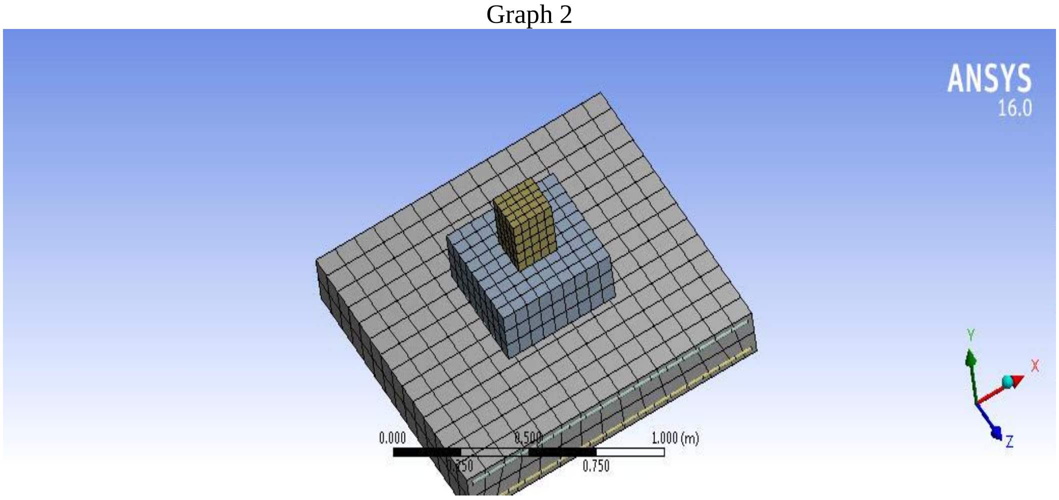


Fig 7

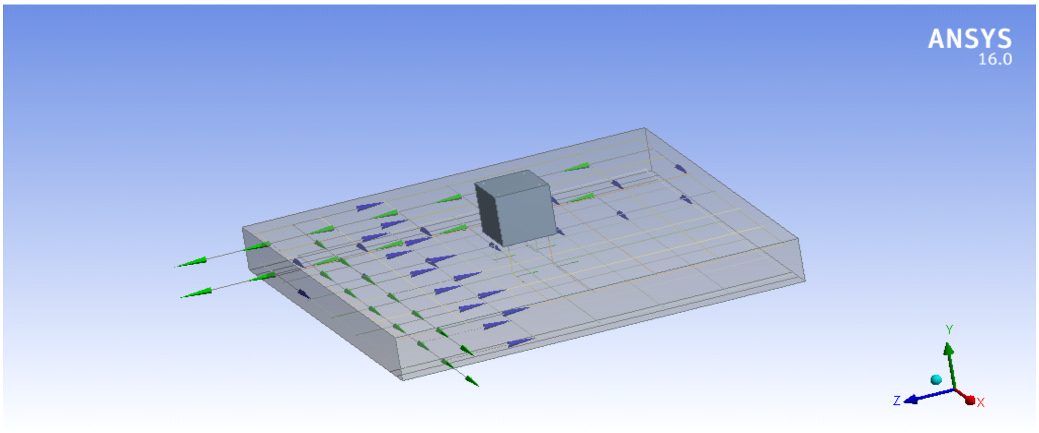


Fig. 8

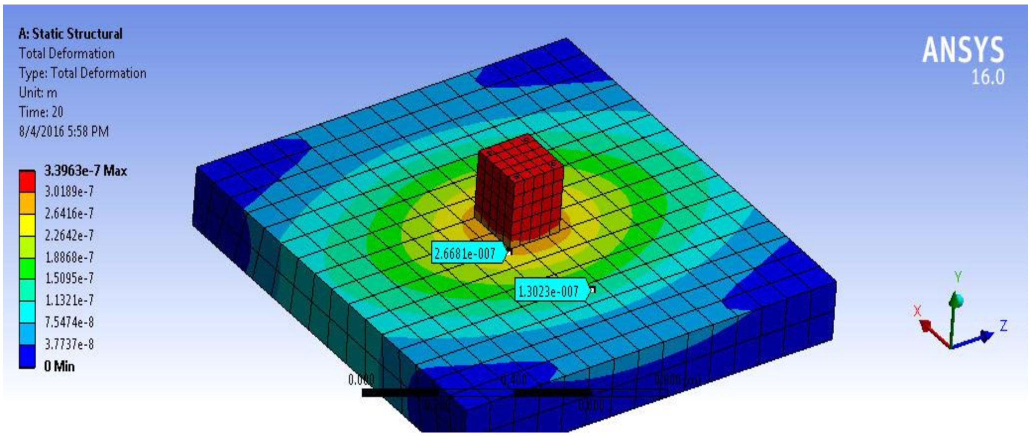


Fig 9

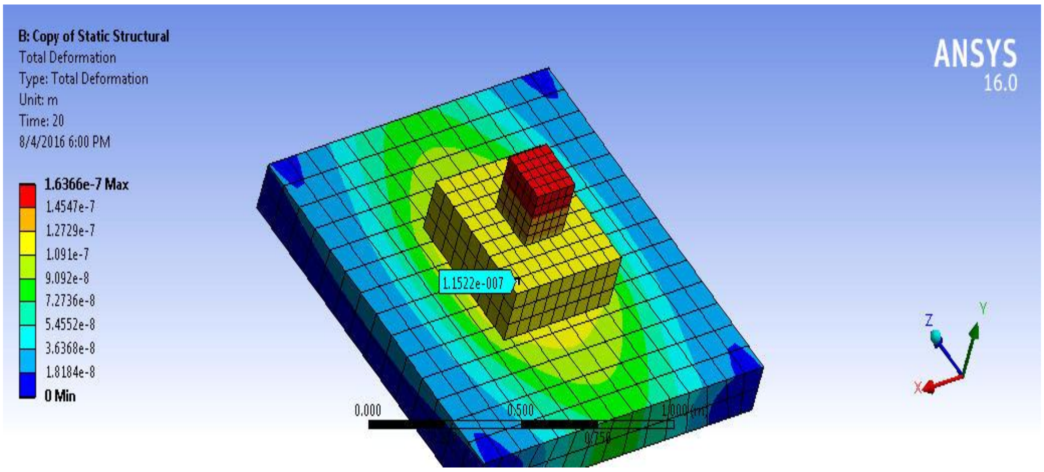


Fig 10

V. Result and Discussion:

In this paper finite element modelling of Flat slab with and without drop panel is conducted. First stage of this project include validation of model with experimental result (5).later on in second stage a drop panel is inserted at critical depth d/2. It is observed that the deformation of slab is considerably reduced. For future work scope research work can be done in shape such as circular, hexagonal, pentagon with various depths. The above study indicates the validation of IS 456:2000.

References:

[1] Ahmed Ibrahim, Salah E.E. El-Metwally, Hamed H. Asker,Mohamed A. El Zareef (2015).Effect of mid-thickness rebar mesh on the behavior and punching shear strength of interior slab–column connection.HBRC Journal (2015).
[2] ANSYS. ANSYS User's Manual Revision 16, ANSYS.
[3] IS 1893-Part 1. Indian Standard Criteria for Earthquake Resistant Design of Structures, Bureau of Indian Standards, New Delhi, India, 2002.
[4] IS 456. Indian Standard Plain and Reinforced Concrete Code of Practice, Bureau of Indian Standards, New Delhi, India, 2000.
[5] M. Priya, S. Greeshma and K. N. Suganya (2015), Finite Element Analysis of Slab - Column Joint Under Lateral Loading .Asian Journal of Civil Engineering (BHRC) vol. 16, no. 2 (2015)
[6] S.R. Hoseini Vaez, A. Kheyroddin, and H. Naderpour(2008),3D Finite Element Simulation of Slab-Column Connections Strengthened with CFRP.*Fourth International Conference on FRP Composites in Civil Engineering (CICE2008)*.