FINITE ELEMENT ANALYSIS OF REINFORCED CONCRETE BEAM SUBJECT TO PURE TORSION

BADIOZAMAN AB. KHALIK

SEPTEMBER 2001

DEPARTMENT OF CIVIL AND CONSTRUCTION
UMIST

FINITE ELEMENT ANALYSIS OF REINFORCED CONCRETE BEAM SUBJECT TO PURE TORSION

by

BADIOEZAMAN AB. KHALIK

A dissertation submitted to the University of Manchester Institute of Science and Technology for the degree of MSc in Structural Engineering

SEPTEMBER 2001

DEPARTMENT OF CIVIL AND CONSTRUCTION

UMIST

ABSTRACT

In the past decades, many experiments were carried out on the concrete subject to torsion but not much on the finite element analysis. In this dissertation, a reinforced concrete beam subject to pure torsion will be modelled using finite element software called ABAQUS. The results obtained from the finite element analysis will be calibrated to the past-published experimental results by Hsu [4] on one of the specimen named B3. A parametric study on the model with various spacing of stirrups is done and the results are compared to hand calculations using the EC 2, BS 8110 and ACI 318-95 design codes.

CHAPTER 1

INTRODUCTION

1.1 OVERVIEW

In the past decades, torsion was not generally taken into account in the design of concrete structures. It was assumed that the torsional effects were minor and could be taken care by the high factor of safety used in the structural design. Now that flexural design of structural concrete has been refined and factors of safety have been reduced, it is necessary to take torsion into account explicitly in design where torsional moments are significant such as curved beams and spiral staircase.

In the early days, the analysis of torsion in concrete members has been based on either classical elastic theory or the plastic theory but later on new theoretical approaches have been proposed as a result of intensified research by many investigators. The new theories were then being compared to experimental results to validate them. Unfortunately, not much analytical studies were carried out to determine the displacements, internal forces, stresses and deformation of reinforced concrete structures due to its complicated nature such as the following:

a) The structural system is composed of two materials, concrete and steel.

b) The concrete is a composite of aggregate, cement paste, water and void. The stress strain relationship for concrete is nonlinear and is a function of many variables. It is difficult to establish useful constitutive relationships and failure criteria valid for multiaxial stress states.

c) Concrete exhibits progressive cracking under increasing load.

d) Concrete deformations are influenced by creep and shrinkage and depend on load and environmental history.

e) The bond slip between the reinforcement and the concrete is difficult to model.

Due to these complexities, engineers in the past have been forced to rely to a great extent on empirical formulas. However, with the advent of digital computers and powerful modern numerical analysis methods such as finite element method, analytical studies of the response of reinforced concrete structures have been able to develop. In this dissertation, finite element analysis software called ABAQUS is used to model a reinforced concrete beam subject to pure torsion.

1.1 OBJECTIVE OF THE STUDY

The objective of the study is to get familiar with ABAQUS and to model a reinforced concrete beam subject to pure torsion. Different type of element such as the beam and solid elements, and meshing will be considered that is suitable for the modelling. The numerical output will be analysed and compared with the previously published experimental results. The finite element model will be calibrated by this comparison. Once the calibration of the model has been validated, or at least to some extent, the parametric studies on the model will be carried out by varying the spacing of stirrups, amount of reinforcement, size of beams etc. The number of parametric studies to be conducted will depend on the availability of time.