CHAPTER 5

FINITE ELEMENT ANALYSIS

5.1 GENERAL

Structural modeling is the process of creation of idealized and simplified representation of structural behaviour and it is an essential step in structural analysis and design. Errors and inadequacies in modeling may cause serious design defects and difficulties. Standard approach in structural design requires use of CASA (Computer Aided Structural Analysis) software with sophisticated options of complete analysis of structural response (displacements and stresses) for different actions (influences and forces).

Sophisticated options include possibilities of application of complex models for more realistic simulation of structural behaviour. The goal is a safe, efficient and rational structural design. Due to great possibilities of software implementation (automatic FE mesh creation, the formation and solving of constitutive equations, numerical and graphic presentation of the results, etc.) Finite Element Method (FEM) has become a dominant method of numerical modeling of complex structural problems. The algorithm shown in Figure 5.1 describes the FEM modeling process in structural analysis.

Basically, FEM modeling includes discretization, approximation and solution phase. Discretization (modeling of geometry/topology i.e. generation of a FE mesh) is the initial phase of FEM modeling process. Errors of discretization may occur due to difference between real topology of the
structure and FE system topology caused by application of unsuitable FE shape or insufficient number of FEs.

Figure 5.1 Finite Element Modeling Process

Next step in FEM modeling is the approximation, i.e. choice of FE from the library of the software. This phase is the numerical modeling in strict sense and comprehends modeling of material behaviour, boundary and interface conditions and some particular phenomena which are important for knowledge of structural response under various types of action.

Most applications of the FE method are concerned with solutions to linear problems. However, the non-linear techniques have become increasingly important, because most of the problems, when properly stated, are in fact non-linear and understanding of the non-linear characteristics are essential for a successful analysis. In structural analysis, it becomes necessary to study the complete non-linear behaviour up to collapse in order to assess the safety of the structure.

The cost incurred and the time consumed for the fabrication and testing of components to understand the behaviour renders finite element
analysis as an economical alternative to many engineering problems. In addition to this, finite element analysis has been a tool of choice for optimizing new designs, verifying the fitness of existing facilities and evaluating new concepts.

A number of finite element modeling software packages are available for solving both linear and non-linear structural analysis problems. The main objective of developing the present accurate FE model is to examine the behaviour of concrete-filled steel tubes under cyclic loading. The details of this behaviour have not been fully clarified because of the lack of methods to evaluate quantitatively the multi-axial stress and strain distributions in steel tube and in-filled concrete as well as the interface action.

In this study, finite element models were developed using the software package ABAQUS version 6.5 (2005). Failure loads and failure modes of concrete-filled steel columns have been studied and compared with the experimental results. The steps involved in the finite element analysis are discussed in detail in this chapter.

5.2 MATERIAL MODELLING

5.2.1 Choice of Element Type

ABAQUS has several element types suitable for numerical analysis: solid two and three dimensional elements, membrane and truss elements, beam elements, and shell elements. The major aim of the analysis is to predict the formation of inelastic local instabilities in a cross-section and the corresponding rotation capacity. Beam, membrane and truss elements are not appropriate for the buckling problem. Solid three dimensional elements (“brick” elements) may be suitable, but the solid elements have only translation degrees of freedom at each node, and require a fine mesh to model regions of high curvature. A finer mesh does not necessarily imply more total
degrees of freedom, as one must consider the number of elements and the degrees of freedom of each element. The most appropriate element type is the shell element. ABAQUS has “thick” and “thin” shells. “Thick” shells should be used in applications where the shell thickness is more than 1/15\textsuperscript{th} of the characteristic length on the surface of the shell.

5.2.2  Modeling of Steel Shell

To analyze the local buckling behaviour, the geometrically non-linear 4-node thick shell element S4R is used to model the thin-walled steel tubes. The accuracy of S4R element with the 3-surface cyclic plasticity model to compute the hysteretic behaviour of stiffened rectangular hollow steel columns was confirmed by Goto et al. (2007). In this study S4R element with six degrees of freedom at each node has been used to model the outer steel shell.

5.2.3  Modeling of In-filled Concrete

In conventional concrete models, multi-axial behaviour under compressive stresses is represented by the plasticity model, while the behaviour under tensile and small compressive stresses is expressed by smeared cracking model (Chen 1982). This model, however, often encounters numerical instability under cyclic load. To circumvent this situation, the concrete damaged plasticity model (three - dimensional 8 noded solid element C3D8R with six degrees of freedom at each node. This damaged plasticity model includes the compressive strength enhancement of concrete due to confinement as well as the post-peak softening behaviour due to concrete crushing. However, compared with the conventional smeared cracking model, the damaged plasticity model is approximate in terms of its tensile behaviour because an isotropic plasticity is assumed in the tension side similar to the compression side.
5.2.4 Modelling of Steel Plates

Steel plates were used to distribute the load over the column section. The loading plates attached to both ends of the column were modelled as 3-dimensional brick elements, type C3D8 (8 noded linear brick) with six degrees of freedom at each node. The steel tube was joined to the loading plates only by the weld elements. The weld between the steel tube and the loading plate was simulated using element type C3D6 (6 noded linear triangular prism) with six degrees of freedom at each node.

5.2.5 Modelling of Steel Concrete Interface

In order to simulate the interface action at the steel concrete interface, two types of contact models were used. In the case of thin-walled in-filled columns, the full use of the accurate hard contact pair model considerably impairs the efficiency of analysis due to numerical instability. Therefore, a mixed use of the hard contact model and the contact spring model is appropriate in modeling the interfaces of in-filled columns to enhance the efficiency of numerical analysis without losing accuracy.

The contact spring model is applied to the interfaces where slip or separation of the facing surfaces is small. The hard contact pair model is applied to the interface between in-filled concrete and steel panels where a large separation of facing surfaces may occur due to local buckling deformation of the outer tubes.

5.3 MESH SIZE

The mesh size of FE model has a big influence on the computed softening behaviour of concrete. It is desirable to minimize the number of
elements within a model, provided the results of the analysis are not unduly affected by removing the elements. In this analysis an aspect ratio of 5 was used for mesh refinement. In total, the model used in the analysis of concrete-filled columns consists of 2952 shell elements and 6080 solid elements.

### 5.4 MATERIAL PROPERTIES

#### 5.4.1 Steel

The material properties obtained from the standard coupon test were input to the ABAQUS model as a set of points on the stress-strain curve. ABAQUS uses true stress and true strain, and hence the values of engineering stress and engineering strain from the standard coupon test were modified before being inserted into the model using the following equations:

\[
\sigma = \sigma (1 + e) \quad (5.1)
\]

\[e = \ln(1 + e) - \frac{\sigma}{E} \quad (5.2)\]

where,

- \(\sigma\) - True Stress
- \(e\) - Engineering Stress

The constitutive stress-strain model for steel used in the analysis is shown in figure 5.2 (a).

#### 5.4.2 Concrete

Concrete material parameters required to define the damaged plasticity model are Young’s modulus of concrete \(E_c = 5000 \sqrt{f_{ck}}\), Poisson’s ratio \(\nu_c = 0.2\) and tensile strength \(f_{ck}/10\). For the remaining parameters the
default values given in ABAQUS were used. The constitutive model for concrete is shown in figure 5.2 (b).

![Figure 5.2 Material Constitutive Models](image)

a) Steel Constitutive Model

b) Concrete Constitutive Model

Figure 5.2 Material Constitutive Models
5.4.3 Constants for Interface Modeling

The friction coefficient ($\mu$) for the Coulomb friction model has been taken as 0.2 for steel-concrete interface (Johansson and Gylltoft 2001) and 1.0 for concrete-to-concrete crack surface. These values have been used both for the hard contact pair model and the contact spring model. In the contact spring model, the elastic constants for contact springs and for shear springs have been adopted as $100 \ E_c L_{min}$ and 2764 kPa/cm respectively (Goto et al 2009). $L_{min}$ is the minimum length of concrete elements at the corresponding interfaces.

The lower part of the columns where large local buckling deformation may occur is discretized into very small square shell elements with aspect ratio 5, while the upper part is modeled by the elastic Timoshenko beam element (ABAQUS B31).

At the boundary between the upper part and the lower part of the in-filled columns, multiple nodes of shell elements are connected to a single node of a beam element by the Multi-Point Constraint (MPC) condition where the displacements of the nodes of shell elements satisfy the assumptions of Timoshenko beam theory at the interface. The FE model for specimen is schematically shown in Figure 5.2.

5.5 BOUNDARY CONDITIONS

The end boundary conditions for all the finite element models were chosen to simulate the actual experimental set up of cantilever type columns. At the immovable end, all the nodes were constrained against translation and at the movable end translation of nodes in the Z and X directions were permitted.
5.6 SIMULATION OF CYCLIC LOADS

In the cyclic loading test, the unidirectional alternating horizontal displacement was quasi-statically applied at the top of the specimens keeping the vertical compressive load constant. The amplitude of the applied horizontal displacement was increased after one cycle of loading similar to the loading pattern adopted in the experimental program.

Figure 5.3 Finite Element Model