Chapter 5

Finite Element Program

An existing finite element (FE) program, written by John F. Hall, is adapted for use with this model. The original program was a linearly elastic, static analysis FE program for solid and structural mechanics. It was designed to solve the equation [K]{x} = {f} for {x} given a linear elastic stiffness matrix and constant force values. It contained only linear springs and four node isotropic plane stress or strain elements.

Modifications of the program are necessary to model reinforced concrete columns. The first step is to add or modify elements to be used for representing reinforced concrete columns confined by steel rebar or fiber reinforced polymer (FRP) sheets. Capabilities for six node tetrahedral and eight node rectangular brick elements are added to the existing program; both using an eight point integration scheme. Each node of the solid element has three translational degrees of freedom. The solid elements are to be used for modeling the concrete. The material model defined in this thesis is implemented into the newly added solid elements. Three material parameters need to be provided as input to these solid elements: the modulus of elasticity, E; Poisson's ratio, ν; and the unconfined compressive strength, f_c'. Further, the linear spring element is modified to be bilinear elastic. The loading behavior of the spring element is bilinear, and the unloading behavior is along the current stiffness value, not the initial stiffness value. Four input quantities are required for the spring elements: the initial modulus of elasticity, E; area, A; yield stress, f_y; and the yield ratio, defined as the ratio of the post yield stiffness to the initial elastic stiffness. The solid and spring elements are used to model a reinforced concrete column with two

different types of confinement materials in subsequent chapters.

Nonlinear solution capabilities and coordinate updating are also added to the program. The applied load is divided into smaller steps so that the full stress versus strain behavior of the material can be mapped. For each small load increment, the program iterates using the elastic stiffness matrix until convergence is achieved. For each iteration, the current difference between the applied load and the internal resisting load is calculated. This force residual is used in conjunction with the elastic stiffness matrix to calculate a new increment in displacement. This increment is added to the previous increments determined during iteration on the current load step and then applied to the previously converged step for calculation of the current internal load. At the end of each converged load increment, the coordinates are updated. The full Fortran code is shown in Appendix E.

To reduce the problem size, only a horizontal slice of the column is considered. Figure 5.1 shows a slice of a typical cylindrical column. The slice is meshed by one
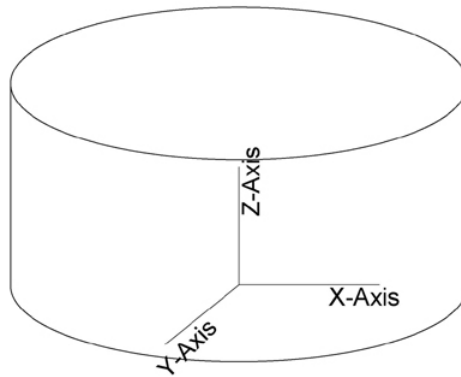

Figure 5.1: Typical slice of column used for FE model.

layer of solid elements. The height of the slice should be equal to half of the hoop spacing or spiral pitch for the case of a steel rebar confined section. Where adequate symmetry exists in the cross sectional shape and rebar layout, only a fraction of the cross section is modeled. For the case of pure axial loading, only one quarter of the slice is typically modeled. When moments are applied to the section, one half of the section is modeled. In both load cases, symmetric boundary conditions are applied. For circular cross sections, the centermost elements of the section are six node solid

elements. All other elements in the section are eight node elements. A typical mesh used for a load case including moments about the x-axis is shown in Figure 5.2. The
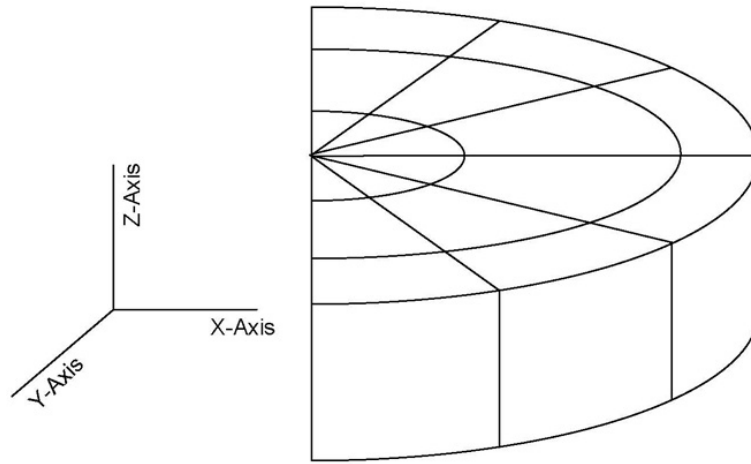

Figure 5.2: Typical mesh used for circular cross sections.

nodes on the left hand face would be restrained in the x-direction. For the case of a square cross section, only eight node solid elements are used and the section is meshed with a simple grid.

The assumption that plane sections remain plane is designed into the program. The nodes in the bottom plane are fixed in the vertical degree of freedom; this vertical direction is labeled as the z-axis in Figures 5.1 and 5.2. The top plane of nodes is defined to move as a plane having three degrees of freedom, representing a vertical displacement and two rotations. In the figures, all nodes in the top plane would share one z degree of freedom, one rotation about the x-axis, and one rotation about the y-axis as a plane. Each top plane node is free to move within the plane that has its orientation defined by the three shared degrees of freedom. These conditions constrain the top and bottom planes to remain planar, a typical assumption for columns.

Since the confinement material is to be explicitly included, a method for modeling steel rebar and FRP sheets must be defined. For steel rebar, spring elements are defined within the bottom layer of nodes to represent the steel hoops. Spirals are modeled in an identical fashion, ignoring the fact that the rebar would change over the height of the given section. No attempt is made to account for the difference in

behavior between a hoop and a spiral. The slice is defined with a height equal to half of the hoop spacing or spiral pitch. Longitudinal rebar is also modeled using bilinear spring elements. In both cases, the post-yield stiffness of the rebar is set to zero. Thus, strain hardening in the rebar is not modeled. For the case of FRP confinement, springs are defined around the top and bottom layers of perimeter nodes to represent the FRP sheet. In this case, the slice height is typically set equal to the height of the steel confined column to which the FRP confined column is being compared. The FRP sheets are represented as purely elastic. Thus, the yield ratio is defined as one. For comparisons between the FE model and experimental results performed in this thesis, the yield stress, f_y, is defined as the rupture stress of the FRP, and the program is terminated if any FRP element reaches its f_y value. The area of the spring element is defined as the thickness of the sheet multiplied by half the slice height. This is the area of the FRP sheet being represented by the individual spring element.

Comparable steel hoop and FRP confined meshes are shown in Figure 5.3. This mesh would be used for axial load, as it represents only one quarter of the cross sectional slice. Thick lines indicate the presence of spring elements in the mesh. All
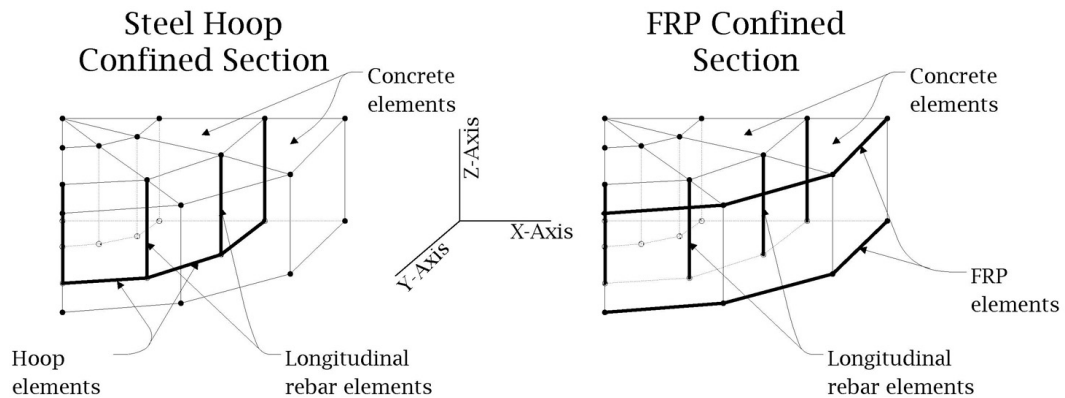

Figure 5.3: Comparable meshes with steel hoops versus FRP confinement.

finite element meshes used for Chapters 6 and 7 are shown in Appendices G and H, respectively.

In the scope of this thesis, several different load configurations are considered. Finite element theory allows for the application of loads in many different ways.

Thus, some details are necessary about how the loads are applied. Axial loads are applied by a force on the z degree of freedom of the top plane. Moments are applied to the x or y rotational degrees of freedom of the top plane. For the case of combined axial load and moment, the member is loaded to the specified axial load, and then the moment is increased to failure. When active confinement is modeled (for example, a cylinder immersed in a pressurized fluid), radial forces are applied to the external nodes in proportion to the area represented by that node. This is not the only way of applying these load cases, but simply the manner chosen for this thesis. For all cases, when the concrete stress is given, this stress is calculated as the weighted average over all integration points lying within the confined area of the section. The weights are given by the product of the Gauss-Legendre weight and the determinant of the Jacobian calculated at that integration point.