7.

BOUNDARY CONDITIONS AND GENERAL CONSTRAINTS

The Specification of Known Joint Displacements Reduces the Number of Equations to be Solved

7.1 INTRODUCTION

The fundamentals of structural analysis and mechanics as applied to the linear static analysis have been summarized in the first several chapters of this book. However, additional computational and modeling techniques used to solve special problems remain to be presented.

It has been established that the displacement method, where the joint displacements and rotations are the unknowns, generates a system of joint equilibrium equations. Both statically determinate and statically indeterminate structures are solved by the displacement method. The global stiffness matrix is the sum of element stiffness matrices and can be formed with respect to all possible joint displacement degrees of freedom. The minimum number of supports required for a stable system is that which will prevent rigid body movement of the structure.

There are several reasons that the general displacement method is not used for non-computer calculations. For most problems, the solution of a large number of equations is required. Also, to avoid numerical problems, a large number of significant figures is required if both bending and axial deformations are included in the analysis of frame structures. One notes that the two traditional displacement analysis methods, moment distribution and slope-deflection, involve only moments and rotations. When those traditional displacement methods are extended to more general frame-type structures, it is necessary to set the axial deformations to zero; which, in modern terminology, is the application of a displacement constraint.

It has been shown that for the development of finite element stiffness matrices it is necessary to introduce approximate displacement shape functions. Based on the same shape functions, it is possible to develop constraints between different coarse and fine finite element meshes in two and three dimensions.

7.2 DISPLACEMENT BOUNDARY CONDITIONS

One of the significant advantages of the displacement method is the ease in specifying displacement boundary conditions. Consider the following set of *N* equilibrium equations formed including the displacements associated with the supports:

$$\mathbf{Ku} = \mathbf{R} \quad \text{Or, in subscript notation} \quad \sum_{j=1}^{N} K_{ij} u_j = R_i \quad i = 1,...N \tag{7.1}$$

If a particular displacement u_n is known and is specified, the corresponding load, or reaction R_n, is unknown. Hence, the *N-1* equilibrium equations are written as:

$$\begin{aligned} \sum_{j=1}^{n-1} K_{ij} u_j &= R_i - K_{in} u_n \quad i = 1,...n-1 \\ \sum_{j=n+1}^{N} K_{ij} u_j &= R_i - K_{in} u_n \quad i = n+1,...N \end{aligned} \quad \text{or,} \quad \overline{\mathbf{K}}\overline{\mathbf{u}} = \overline{\mathbf{R}} \tag{7.2}$$

This simple modification to the stiffness and load matrices is applied to each specified displacement and the *nth* row and column are discarded. For a fixed support, where the displacement is zero, the load vectors are not modified. Those modifications, resulting from applied displacements, can be applied at the element level, before formation of the global stiffness matrix. After all displacements have been calculated, the load associated with the specified displacements can be calculated from the discarded equilibrium equation. This same basic approach can be used where the displacements are specified as a function of time.

It should be apparent that ***it is not possible to specify both*** u_n ***and*** R_n at the same degree of freedom. One can design a structure so that a specified displacement will result from a specified load; therefore, ***this is a structural design problem*** and not a problem in structural analysis.

7.3 NUMERICAL PROBLEMS IN STRUCTURAL ANALYSIS

Many engineers use large values for element properties when modeling rigid parts of structures. This can cause large errors in the results for static and dynamic analysis problems. In the case of nonlinear analysis the practice of using unrealistically large numbers can cause slow convergence and result in long computer execution times. Therefore, the purpose of this section is to explain the physical reasons for those problems and to present some guidelines for the selection of properties for stiff elements.

Elements with infinite stiffness and rigid supports do not exist in real structures. We can only say that an element, or a support, is stiff relative to other parts of the structure. In many cases, the relative stiffness of what we call a rigid element is 10 to 1,000 times the stiffness of the adjacent flexible elements. The use of these realistic values will not normally cause numerical problems in the analysis of the computer model of a structure. However, if a relative value of 10^{20} is used, a solution may not be possible, because of what is known as ***truncation errors***.

To illustrate truncation errors, consider the simple three-element model shown in Figure 7.1.

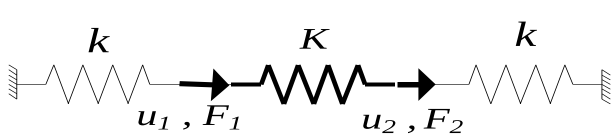

Figure 7.1 Example to Illustrate Numerical Problems

The equilibrium equations for this simple structure, written in matrix form, are the following:

$$\begin{bmatrix} K+k & -K \\ -K & K+k \end{bmatrix} \begin{bmatrix} u_1 \\ u_2 \end{bmatrix} = \begin{bmatrix} F_1 \\ F_2 \end{bmatrix} \tag{7.3}$$

Most structural analysis programs are written in double precision, and the stiffness terms have approximately 15 significant figures and can be in the range of 10^{-308} to 10^{+308}. Therefore, if the stiff element has a stiffness of K=10^{20} k, the term K+k is truncated to K and the equilibrium equations are singular and cannot be solved. If K=10^{12} k, approximately 12 significant figures are lost and the solution is accurate to approximately three significant figures. The equation solvers used in the well-written structural analysis programs can sometimes detect this type of error and warn the user. However, for large systems, this type of error can be cumulative and is not always detected by the computer program.

This problem can be avoided by using realistic stiffness values, or by using constraints in the place of very stiff elements. This is one reason the rigid floor diaphragm constraint is often used in the solution of multistory buildings, because the in-plane stiffness of the floor system is often several orders-of-magnitude greater than the bending stiffness of the columns that connect the stiff floor slabs.

In nonlinear dynamic analysis, iteration is often used to satisfy equilibrium at the end of each time step. If elements have a large stiffness change during the time step, the solution can oscillate about the converged solution for alternate iterations. To avoid this convergence problem, it is necessary to select realistic stiffness values; or displacement constraints can be activated and deactivated during the incremental solution.

7.4 GENERAL THEORY ASSOCIATED WITH CONSTRAINTS

Structural engineers have used displacement constraints in structural analysis for over a century. For example, the two dimensional portal frame shown in Figure 7.2 has six *displacement* degrees of freedom (DOF). Therefore, six independent joint loads are possible.

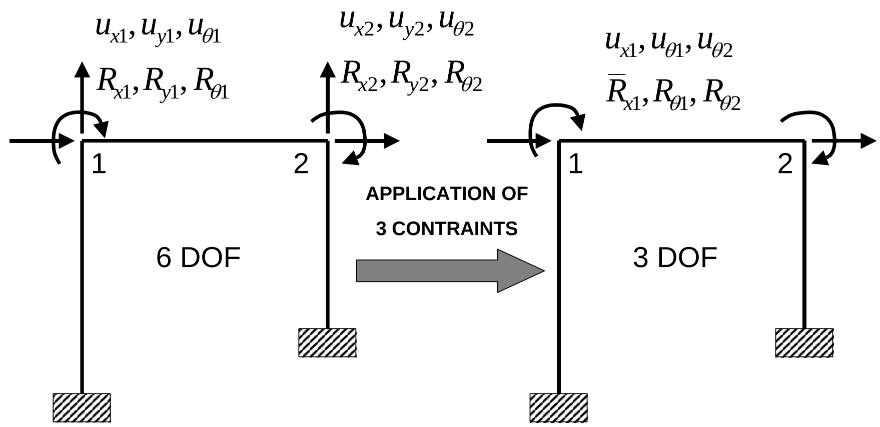

Figure 7.2 Utilization of Displacement Constraints in Portal Frame Analysis

Using hand calculations and the slope-deflection method, it is common practice to neglect axial deformations within the three members of the portal frame. In mathematical notation, those three constraint equations can be written as:

$$
\begin{aligned}
u_{y1} &= 0 \\
u_{y2} &= 0 \\
u_{x2} &= u_{x1}
\end{aligned}
\tag{7.4}
$$

As a result of these constraints, the following load assumptions must be made:

$$
\begin{aligned}
R_{y1} &= 0 \\
R_{y2} &= 0 \\
\overline{R}_{x1} &= R_{x1} + R_{x2}
\end{aligned}
\tag{7.5}
$$

Note the similarities between the displacement compatibility conditions, Equation (7.4), and the force equilibrium requirements, Equation (7.5).

From this simple example, the following general comments can be made:

1. The application of a constraint equation must be justified by a physical understanding of structural behavior. In this case, we can say that the axial deformations are small compared to lateral deformation u_{x1}. Also, the axial deformations in the columns do not cause significant bending forces within the other members of the structure. In addition, vertical loads cannot be applied that can cause horizontal displacements in the real structure.

2. In general, for each application of a constraint equation, one global joint displacement degree of freedom is eliminated.

3. The force association with each axial deformation, which has been set to zero, cannot be calculated directly. Because the axial deformation has been set to zero, a computer program based on a displacement method will produce a zero axial force. This approximation can have serious consequences if “automatic code design checks” are conducted by the computer program.

4. The constraint equations should be applied at the element stiffness level before addition of element stiffness matrices to the global joint equilibrium equations.

7.5 FLOOR DIAPHRAGM CONSTRAINTS

Many automated structural analysis computer programs use master-slave constraint options. However, in many cases the user’s manual does not clearly define the mathematical constraint equations that are used within the program. To illustrate the various forms that this constraint option can take, let us consider the floor diaphragm system shown in Figure 7.3.

The diaphragm, or the physical floor system in the real structure, can have any number of columns and beams connected to it. At the end of each member, at the diaphragm level, six degrees of freedom exist for a three-dimensional structure before introduction of constraints.

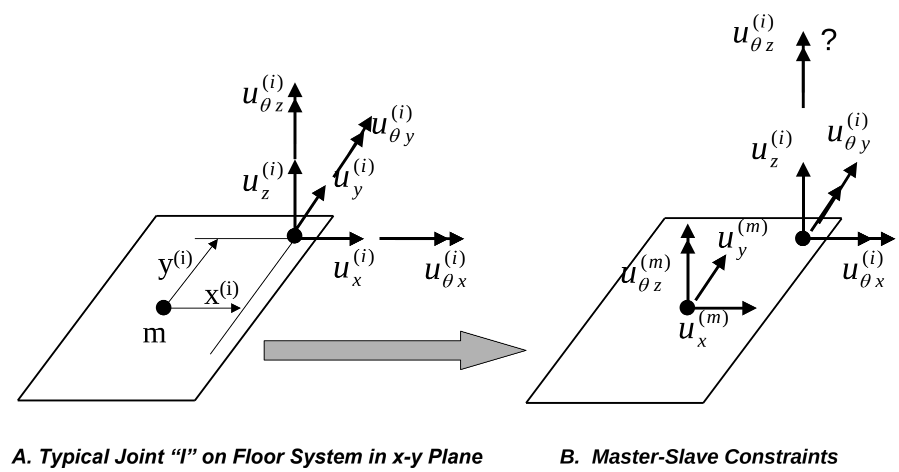

Figure 7.3 Rigid Diaphragm Approximation

Field measurements have verified for a large number of building-type structures that the in-plane deformations in the floor systems are small compared to the inter-story horizontal displacements. Hence, it has become common practice to assume that the in-plane motion of all points on the floor diaphragm move as a rigid body. Therefore, the in-plane displacements of the diaphragm can be expressed in terms of two displacements, $u_x^{(m)}$ and $u_y^{(m)}$, and a rotation about the z-axis, $u_{z\theta}^{(m)}$.

In the case of static loading, the location of the master node (m) can be at any location on the diaphragm. However, for the case of dynamic earthquake loading, ***the master node must be located at the center of mass*** of each floor if a diagonal mass matrix is to be used. The SAP2000 program automatically calculates the location of the master node based on the center of mass of the constraint nodes.

As a result of this rigid diaphragm approximation, the following compatibility equations must be satisfied for joints attached to the diaphragm:

$$u_x^{(i)} = u_x^{(m)} - y^{(i)} u_{\theta z}^{(m)}$$
$$u_y^{(i)} = u_y^{(m)} + x^{(i)} u_{\theta z}^{(m)} \tag{7.6}$$

The rotation $u_{\theta z}^{(i)}$ may or may not be constrained to the rigid body rotation of the diaphragm. This decision must be based on how the beams and columns are physically connected to the floor system. In the case of a steel structure, the structural designer may specify that the floor slab is released in the vicinity of the joint, which would allow the joint to rotate independently of the diaphragm. On the other hand, in the case of a poured-in-place concrete structure, where columns and beams are an intricate part of the floor system, the following additional constraint must be satisfied:

$$u_{\theta z}^{(i)} = u_{\theta z}^{(m)} \tag{7.7}$$

Or in matrix form, the displacement transformation is:

$$\begin{bmatrix} u_x^{(i)} \\ u_y^{(i)} \\ u_{\theta z}^{(i)} \end{bmatrix} = \begin{bmatrix} 1 & 0 & -y^{(i)} \\ 0 & 1 & x^{(i)} \\ 0 & 0 & u_{\theta z}^{(i)} \end{bmatrix} \begin{bmatrix} u_x^{(m)} \\ u_y^{(m)} \\ u_{\theta z}^{(m)} \end{bmatrix} \text{ or, } \mathbf{u}^{(i)} = \mathbf{T}^{(i)} \mathbf{u}^{(m)} \tag{7.8}$$

If displacements are eliminated by the application of constraint equations, the loads associated with those displacements must also be transformed to the master node. From simple statics the loads applied at joint “i” can be moved to the master node “m” by the following equilibrium equations:

$$R_x^{(mi)} = R_x^{(i)}$$
$$R_y^{(mi)} = R_y^{(i)} \tag{7.9}$$
$$R_{\theta z}^{(mi)} = R_{\theta z}^{(i)} - y^{(i)} R_x^{(i)} + x^{(i)} R_y^{(i)}$$

Or in matrix form. the load transformation is:

$$\begin{bmatrix} R_x^{(mi)} \\ R_y^{(mi)} \\ R_{\theta z}^{(mi)} \end{bmatrix} = \begin{bmatrix} 1 & 0 & 0 \\ 0 & 1 & 0 \\ -y^{(i)} & x^{(i)} & 1 \end{bmatrix} \begin{bmatrix} R_x^{(i)} \\ R_y^{(i)} \\ R_{\theta z}^{(i)} \end{bmatrix} \text{ Or, } \mathbf{R}^{(mi)} = \mathbf{T}^{(i)^T} \mathbf{R}^{(i)} \tag{7.10}$$

Again, one notes that the force transformation matrix is the transpose of the displacement transformation matrix.

The total load applied at the master point will be the sum of the contributions from all slave nodes. Or:

$$\mathbf{R}^{(m)} = \sum_i \mathbf{R}^{(mi)} = \sum \mathbf{T}^{(i)^T} \mathbf{R}^{(i)} \tag{7.11}$$

Now, consider a vertical column connected between joint *i* at level m and joint j at level m+1, as shown in Figure 7.4. Note that the location of the master node can be different for each level.

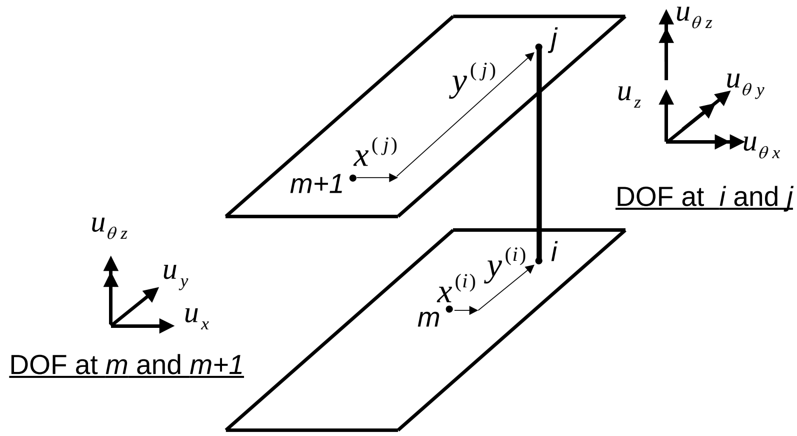

Figure 7.4 Column Connected Between Horizontal Diaphragms

From Equation (7.6) it is apparent that the displacement transformation matrix for the column is given by

$$\begin{bmatrix} u_x^{(i)} \\ u_y^{(i)} \\ u_z^{(i)} \\ u_{\theta x}^{(i)} \\ u_{\theta y}^{(i)} \\ u_{\theta z}^{(i)} \\ u_x^{(j)} \\ u_y^{(j)} \\ u_z^{(j)} \\ u_{\theta x}^{(j)} \\ u_{\theta y}^{(j)} \\ u_{\theta z}^{(j)} \end{bmatrix} = \begin{bmatrix} 1 & 0 & 0 & 0 & 0 & 0 & -y^{(i)} & 0 & 0 & 0 & 0 & 0 & 0 & 0 \\ 0 & 1 & 0 & 0 & 0 & 0 & x^{(i)} & 0 & 0 & 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & 1 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 1 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & 1 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 & 1 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 & 0 & 0 & 1 & 0 & 0 & 0 & 0 & 0 & -y^{(j)} \\ 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 1 & 0 & 0 & 0 & 0 & x^{(j)} \\ 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 1 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 1 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 1 & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 1 & 0 \end{bmatrix} \begin{bmatrix} u_x^{(m)} \\ u_y^{(m)} \\ u_z^{(i)} \\ u_{\theta x}^{(i)} \\ u_{\theta y}^{(i)} \\ u_{\theta z}^{(i)} \\ u_{\theta z}^{(m)} \\ u_x^{(m+1)} \\ u_y^{(m+1)} \\ u_z^{(i)} \\ u_{\theta x}^{(i)} \\ u_{\theta y}^{(i)} \\ u_{\theta z}^{(i)} \\ u_{\theta z}^{(m+1)} \end{bmatrix} \quad (7.12)$$

Or in symbolic form:

$$\mathbf{d} = \mathbf{B}\mathbf{u} \quad (7.13)$$

The displacement transformation matrix is 12 by 14 if the z-rotations are retained as independent displacements. The new 14 by 14 stiffness matrix, with respect to the master and slave reference systems at both levels, is given by:

$$\mathbf{K} = \mathbf{B}^{\mathbf{T}}\mathbf{k}\mathbf{B} \quad (7.14)$$

where $\mathbf{k}$ is the initial 12 by 12 global stiffness matrix for the column. It should be pointed out that the formal matrix multiplication, suggested by Equation (7.14), need not be conducted within a computer program. Sparse matrix operations reduce the numerical effort significantly.

In the case of a beam at a diaphragm level, the axial deformation will be set to zero by the constraints, and the resulting 8 by 8 stiffness matrix will be in reference to six rotations and two vertical displacements. Therefore, the force in the beam element will be zero.

7.6 RIGID CONSTRAINTS

There are several different types of constraints that require displacements at one point to be related to displacements at another point. The most general form of a three-dimensional rigid constraint is illustrated in Figure 7.5.

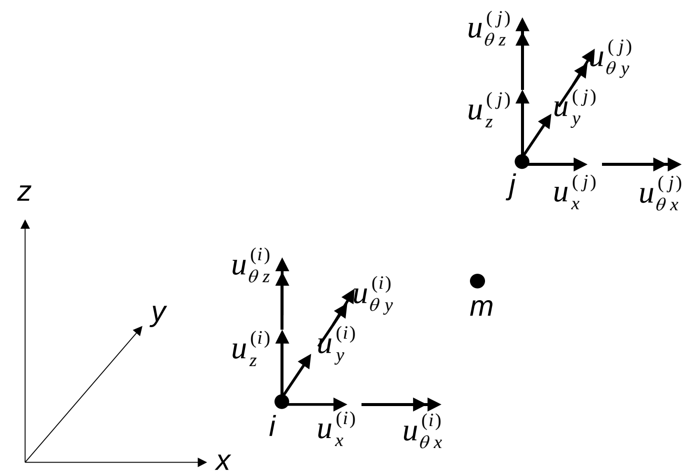

Figure 7.5 Rigid Body Constraints

The points i, j and m are all points on a body that can be considered to move with six rigid body displacements. Any point in space can be considered as the master node for static loading; however, for dynamic analysis, the master node must be at the center of the mass if we wish to restrict our formulation to a diagonal mass matrix.

It is apparent from the fundamental equations of geometry that all points connected to the rigid body are related to the displacements of the master node by the following equations:

$$
\begin{aligned}
u_x^{(i)} &= u_x^{(m)} + (z^{(i)} - z^{(m)})u_{\theta y}^{(m)} - (y^{(i)} - y^{(m)})u_{\theta z}^{(m)} \\
u_y^{(i)} &= u_y^{(m)} - (z^{(i)} - z^{(m)})u_{\theta x}^{(m)} + (x^{(i)} - x^{(m)})u_{\theta z}^{(m)} \\
u_z^{(i)} &= u_z^{(m)} + (y^{(i)} - y^{(m)})u_{\theta x}^{(m)} - (x^{(i)} - x^{(m)})u_{\theta y}^{(m)} \\
u_{\theta x}^{(i)} &= u_{\theta x}^{(m)} \\
u_{\theta y}^{(i)} &= u_{\theta y}^{(m)} \\
u_{\theta z}^{(i)} &= u_{\theta z}^{(m)}
\end{aligned}
\tag{7.15}
$$

The constraint equations for point j are identical to matrix Equation (7.15) with i replaced with j.

7.7 USE OF CONSTRAINTS IN BEAM-SHELL ANALYSIS

An example that illustrates the practical use of a three-dimensional rigid constraint is the beam-slab system shown in Figure 7.6.

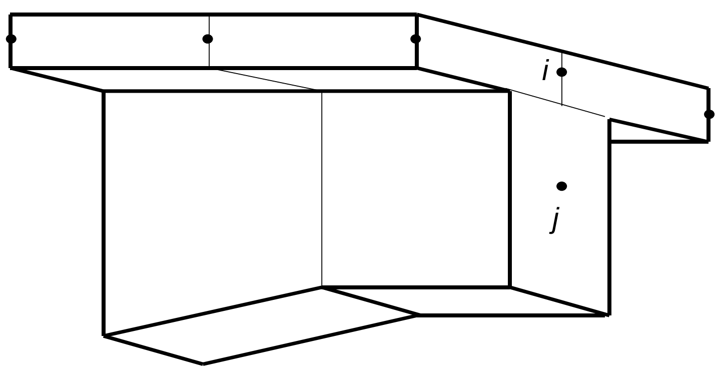

Figure 7.6 Connection of Beam to Slab by Constraints

It is realistic to use four-node shell elements to model the slab and two-node beam elements to model the beam. Both elements have six DOF per node. However, there are no common nodes in space to directly connect the two element types. Therefore, it is logical to connect node i, at the mid-surface of the slab, with point j at the neutral axis of the beam with a rigid constraint. If these constraints are enforced at the shell nodes along the axis of the beam, it will allow the natural interaction of the two element types. In addition to reducing the number of unknowns, it avoids the problem of selecting an effective width of the slab. Also, it allows non-prismatic beams, where the neutral axis in not on a straight line, to be realistically modeled. To maintain compatibility between the beam and slab, it may be necessary to apply the rigid-body constraint at several sections along the axis of the beam.

7.8 USE OF CONSTRAINTS IN SHEAR WALL ANALYSIS

Another area in which the use of constraints has proven useful is in the analysis of perforated concrete shear walls. Consider the two-dimensional shear wall shown in Figure 7.7a.

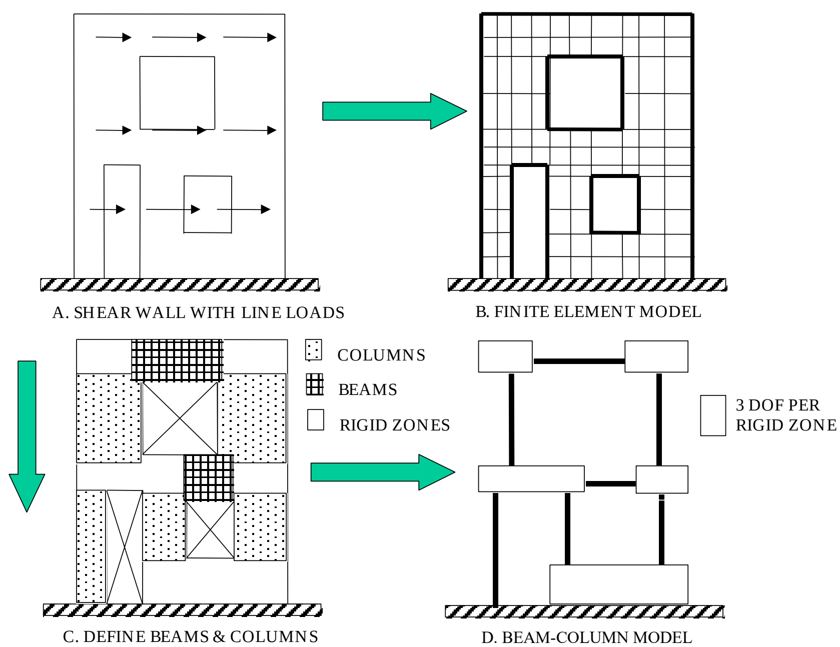

Figure 7.7 Beam-Column Model of Shear Wall

Many engineers believe that the creation of a two-dimensional finite element mesh, as shown in Figure 7.7b, is the best approach to evaluate the displacements and stresses within the shear wall. In the author's opinion, this approach may not be the best for the following reasons:

1. As previously illustrated, the use of four-node plane elements for frame analysis does not accurately model linear bending. The approximation of constant shear stress within each element makes it very difficult to capture the parabolic shear distribution that exists in the classical frame element.

2. If a very fine mesh is used, the linear finite element solution will produce near infinite stresses at the corners of the openings. Because the basic philosophy of reinforced concrete design is based on cracked sections, it is not possible to use the finite element results directly for design.

3. Using common sense and a physical insight into the behavior of the structure, it is possible to use frame elements to create a very simple model that accurately captures the behavior of the structure and directly produces results that can be used to design the concrete elements.

Figure 7.7c illustrates how the shear wall is reduced to a frame element model interconnected with rigid zones. The columns are first defined by identifying regions of the structure that have two stress-free vertical sides. The beams are then defined by identifying areas that have two stress-free horizontal sides. The length of each beam and column should be increased by approximately 20 percent of the depth of the element to allow for deformations near the ends of the elements. The remaining areas of the structure are assumed to be rigid in-plane.

Based on these physical approximations, the simple model, shown in Figure 7.7d, is produced. Each rigid area will have three DOF, two translations and two rotations. The end of the frame elements must be constrained to move with these rigid areas. Therefore, this model has only 12 DOF. Additional nodes within the frame elements may be required to accurately model the lateral loading.

7.9 USE OF CONSTRAINTS FOR MESH TRANSITIONS

It is a fact that rectangular elements are more accurate than arbitrary quadrilateral elements. Also, regular eight-node prisms are more accurate than hexahedral elements of arbitrary shape. Therefore, there is a motivation to use constraints to connect a fine mesh with coarse mesh.

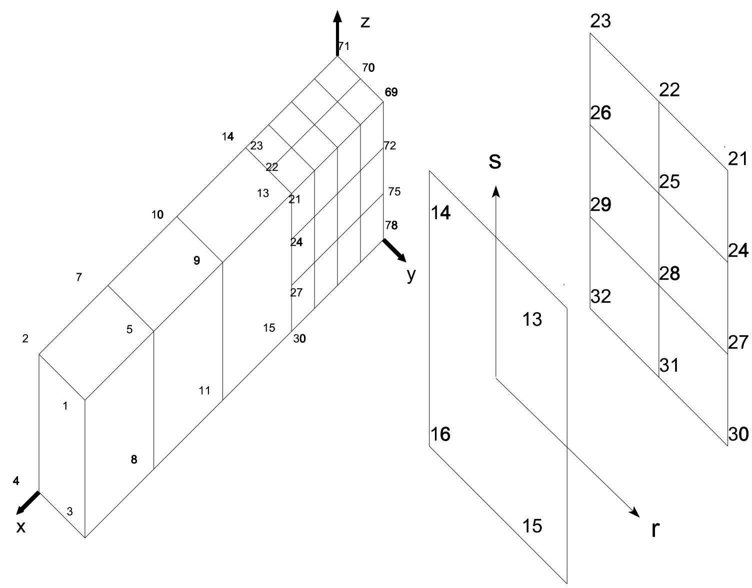

Figure 7.8 Use of Constraints to Merge Different Finite Element Meshes

To illustrate the use of constraints to merge different sized elements, consider the three-dimensional finite element shown in Figure 7.8.

The easiest method to generate the mesh shown in Figure 7.8 is to use completely different numbering systems to generate the coarse and fine mesh areas of the finite element model. The two sections can then be connected by displacement constraints. To satisfy compatibility, it is necessary that ***the fine mesh be constrained to the coarse mesh***. Therefore, the shape functions of the surface of the coarse mesh must be used to evaluate the displacements at the nodes of the fine mesh. In this case, the 36 DOF of the12 fine mesh nodes, numbers 21 to 32, are related to the displacements at nodes 13 to 16 by 36 equations of the following form:

$$u_c = N_{13}u_{13} + N_{14}u_{14} + N_{15}u_{15} + N_{16}u_{16} \tag{7.16}$$

The equation is applied to the x, y and z displacements at the 12 points. The bilinear shape functions, N_i, are evaluated at the natural coordinates of the 12 points. For example, the natural coordinates for node 25 are $r = 0$ and $s = 1/3$. It is apparent that these displacement transformations can automatically be formed and applied within a computer program. This approach has been used in computer programs that use adaptive mesh refinement.

7.10 LAGRANGE MULTIPLIERS AND PENALTY FUNCTIONS

In rigid-body mechanics the classical approach to specify displacement constraints is by using Lagrange multipliers. A more recent approach used in computational mechanics is to use penalty functions, within the variational formulation of the problem, to enforce constraint conditions.

The penalty method can be explained using a simple physical approach in which the constraint is enforced using a ***semi-rigid element***. To illustrate this approach Equation (7.17) can be written as:

$$N_{13}u_{13} + N_{14}u_{14} + N_{15}u_{15} + N_{16}u_{16} - u_c = 0 \approx e \text{ or, } e = \mathbf{B}_c\mathbf{u} \tag{7.17}$$

An equation of this form can be written for all degrees of freedom at the constraint node. The displacement transformation matrix $\mathbf{B}_c$ is a 1 by 5 matrix for each constraint displacement. For the constraint equation to be satisfied, the error e must be zero, or a very small number compared to the other displacements in the equation. This can be accomplished by assigning a large stiffness k_c, or ***penalty term***, to the error in the constraint equation. Hence, the force associated with the constraint is $f_c = k_c e$ and the 5 by 5 constraint element stiffness matrix can be written as:

$$\mathbf{k}_c = \mathbf{B}_c^T k_c \mathbf{B}_c \tag{7.18}$$

As the value of k_c is increased, the error is reduced and the strain energy within the constraint element will approach zero. Therefore, the energy associated with the constraint element can be added directly to the potential energy of the system before application of the principle of minimum potential energy.

It should be pointed out that the penalty term should not be too large, or numerical problems may be introduced, as illustrated in Figure 7.1. This can be avoided if the penalty term is three to four orders-of-magnitude greater than the stiffness of the adjacent elements.

The Lagrange multiplier approach adds the constraint equations to the potential energy. Or:

$$\Omega = \frac{1}{2}\mathbf{u}^{\mathbf{T}}\mathbf{K}\mathbf{u} - \mathbf{u}^{\mathbf{T}}\mathbf{R} + \sum_{j=1}^{J}\lambda_j \mathbf{B}_j \mathbf{u} \tag{7.19}$$

where λ_j is defined as the Lagrange multiplier for the constraint *j*. After the potential energy is minimized with respect to each displacement and each Lagrange multiplier, the following set of equations is produced:

$$\begin{bmatrix} \mathbf{K} & \mathbf{B} \\ \mathbf{B}^T & \mathbf{0} \end{bmatrix} \begin{bmatrix} \mathbf{u} \\ \lambda \end{bmatrix} = \begin{bmatrix} \mathbf{R} \\ \mathbf{0} \end{bmatrix} \tag{7.20}$$

The number of equations to be solved is increased by “J” additional equations. Equation (7.20) has both equilibrium equations and equations of geometry. Also, the symmetric matrix is not positive-definite. Therefore, pivoting may be required during the solution process. Hence, the penalty method is the preferable approach.

7.11 SUMMARY

Traditionally, constraints were used to reduce the number of equations to be solved. At the present time, however, the high speed of the current generation of inexpensive personal computers allows for the double-precision solution of several thousand equations within a few minutes. Hence, constraints should be used to avoid numerical problems and to create a realistic model that accurately predicts the behavior of the real structure.

Constraint equations are necessary to connect different element types together. In addition, they can be very useful in areas of mesh transitions and adaptive mesh refinement.

Care must be exercised to avoid numerical problems if penalty functions are used to enforce constraints. The use of Lagrange multipliers avoids numerical problems; however, additional numerical effort is required to solve the mixed set of equations.