WCCM V
Fifth World Congress on
Computational Mechanics
July 7-12, 2002, Vienna, Austria
Eds.: H.A. Mang, F.G. Rammerstorfer,
J. Eberhardsteiner

Modeling of connections between dissimilar finite element domains

H. Werkle*

Department of Civil Engineering
University of Applied Sciences Konstanz, Brauneggerstr. 55, D-78476 Konstanz, Germany
e-mail: werkle@fh-konstanz.de

Key words: finite element method, finite element modelling, structural analysis, column, deep beams, flat slabs, foundation slabs, soil-structure interaction

Abstract

In structural engineering, domains of finite elements with dissimilar stress assumptions and degrees of freedom are often used in the same model. A typical example are elements for beams and plates in bending or in plane stress in the analysis of flat slabs and shear walls, respectively.

The paper presents a consistent approach for modeling the connection between dissimilar finite element domains. The nodal forces of the source system are represented by their stress pattern. These are transformed into the target system by a linear relationship. Similarly the transformation of the displacements is formulated. Whereas the equilibrium conditions are fulfilled, the compatibility of the displacements are only fulfilled approximately. Using both relationships the stiffness matrix of the source system is transformed into the target system. The method is called EST or Equivalent Stiffness Transformation.

The application of the EST is shown for the beam-plate problem. Examples relating to the column-wall, column-slab and the halfspace-slab problem illustrate the practical value of the method. The EST can be applied for a large class of problems in which dissimilar finite element domains have to be connected.

1 Introduction

In the analysis of structures, finite elements with different stress representations and different degrees of freedom are often used in the same model. A typical example is the connection of beam elements with plane stress finite elements. In the beam element the stresses are summed up to stress resultants as longitudinal forces, shear forces and bending moments. However, only distributed loads are allowed for the plate in order to avoid stress and displacement singularities. In addition the moment communication at the end of the beam element has to be modeled, Fig. 1. Hence, the consistent modeling of the connection of a beam element with plane stress elements is not obvious.

There are many cases where the transition between different stress and displacement systems has to be modeled. A usual problem in reinforced concrete structures is the connection of columns of flat slabs with the slab where the columns are represented by beam elements and the slab by plate elements in bending. Other examples are encountered when modeling soil-structure interaction in foundation slabs.

2 Models for element transitions

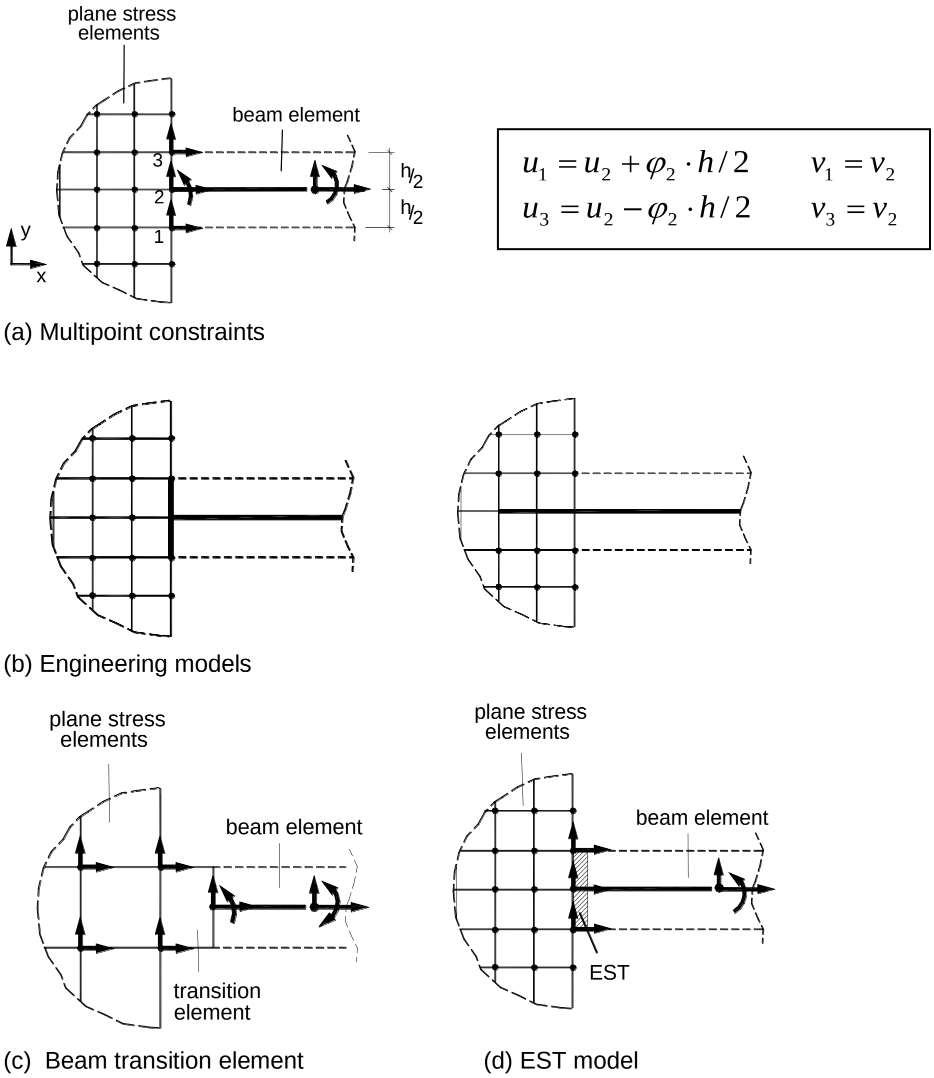
There are various ways to model transitions between different element types. These are

- Multipoint constraints
  - Transformation method
  - Lagrange multiplier method
  - Penalty method
- Transition elements
- Engineering approaches
- Equivalent stiffness transformation (EST)

To demonstrate the different approaches for element transitions the connection of a beam with plane stress elements is considered as example.

For multipoint constraints it is assumed that the displacements of the nodes can be described by rigid elements. For the beam-plate connection in Fig. 1(a), e.g., these rigid constraints can be described for the nodes 1, 2 and 3 as given in the Figure. The degrees of freedom of the 'slave' nodes 1 and 3 are expressed by the translations and the rotation of 'master' node 2. If the displacement constraints are introduces into the global system of equations of the finite element model the degrees of freedom u_1, v_1, u_3 and v_3 can be eliminated. This method is called transformation method.

There are other methods to take into account multipoint constraints. These are the Lagrange multiplier method and the Penalty method. In the Lagrange multiplier method additional variables are introduced into the global equations. They are called Lagrange multipliers and can be regarded as forces of constraint. For each constraint a Lagrange variable is introduced and the global equations are augmented by the corresponding constraint condition as additional equation.



$$u_1 = u_2 + \varphi_2 \cdot h/2 \qquad v_1 = v_2$$
$$u_3 = u_2 - \varphi_2 \cdot h/2 \qquad v_3 = v_2$$

Figure 1: Models for element transitions

In the penalty method additional variables the so-called penalty numbers are introduced. Based on the penalty numbers additional stiffness as well as additional forces are introduced into the global equations. The number of equations is not augmented. However the fulfillment of the constraint conditions is only approximate and depends on the penalty numbers chosen. If penalty numbers are zero the constraints are ignored. As the penalty numbers become large the constraints are very nearly satisfied. However, penalty numbers should not be chosen too large in order to avoid numerical ill-conditioning of the modified global system.

The transformation method as well as the Lagrange multiplier method fulfill the constraint conditions exactly whereas the penalty method is an approximate method. All multipoint constraints introduce artificial rigid elements into the model which may result in stress singularities and other disturbances of stresses and displacements.

Transition elements are another approach to connect dissimilar finite element domains. They are special finite elements to connect two elements of different types. To demonstrate the formulation of a transition element, the simple beam transition element given in Fig. 1(b) is considered. The degrees of freedom of the element include two displacements and a rotation at the side where the beam is connected and two displacements per node where the plane stress element is connected. The transition element is a plane stress element with special interpolation functions which impose the static and kinematic assumptions given by the degrees of freedom of a beam on one side and a plane continuum element on the other side. Transition elements can be used to couple dissimilar elements without the use of rigid constraint equations. They can be applied to one-to-one element connections only. Therefore they are not suited to multiple element connections as e.g. in Fig. 2. Hence, adaptive h-meshing is not possible with transition elements.

For engineering purposes, models as shown in Fig. 1(c) are used. The short beam elements act as rigid constraints. If the stiffness of the beam elements is chosen large enough the model fulfills approximately the a multipoint constraint conditions. However, it should not be chosen to large to avoid numerical ill-conditioning of the global equations. The stiffness in the plate is overestimated by these models. The additional beam elements in the plate may result in disturbances of stresses.

A new approach for element transitions is developed based on the fulfillment of the equilibrium conditions and the approximate compatibility of the displacements between the two elements [1]. It is called Equivalent Stiffness Transformation or EST. The model avoids rigid constraints, allows multiple element connections and hence is suited for adaptive meshing.

3 Equivalent stiffness transformation

3.1 Method of transformation

In the Equivalent Stiffness Transformation (EST) two stress systems, the source system and the target system are considered. The stiffness matrix formulated in the source system is transformed by EST into the target system in which the global equations are formulated. In the example of the beam-plane stress element connection, the source system is the linear stress distribution in the beam which can be summed up to stress resultants. The stiffness matrix of the beam corresponds to the end forces and moments of the beam and the corresponding degrees of freedom. The target system is the plate in

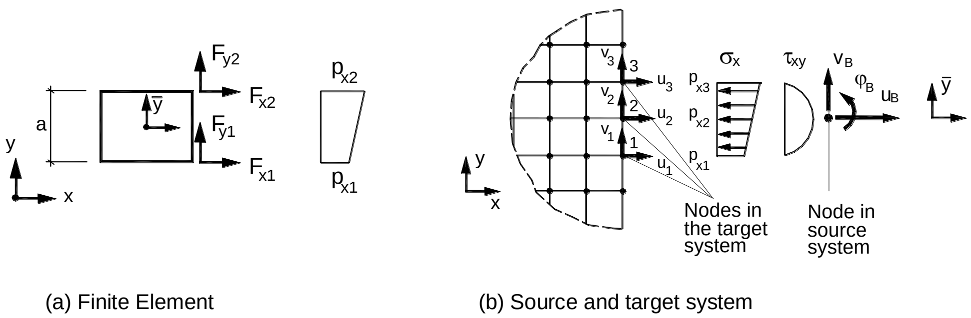


Figure 2: EST model

plane stress. Its degrees of freedom are the nodal displacements of the element. Hence the stiffness matrix of the beam element relating to the degrees of freedom u_B,, v_B and φ_B is to be transformed into the degrees of freedom u_1, v_1 , u_2,, v_2, u_3 and v_3 of the target system, Fig. 2.

The transformation of the stiffness matrix in the source system is done in three steps:

Step 1: Determination of the stresses of the source system at the nodes of the target system

In the source system the stresses are expressed by the nodal forces $\underline{F}_S$. The stresses $\underline{p}_T$ at the nodal points of the target system are

$$\underline{p}_T = \underline{X} \cdot \underline{F}_S \tag{1}$$

For the beam-plate-connection the stresses in x-direction at the nodes 1 to *n* are

$$\begin{bmatrix} p_{x,1} \\ p_{x,2} \\ p_{x,3} \\ \\ p_{x,n_{st}} \end{bmatrix} = \begin{bmatrix} 1 & \bar{y}_1 \\ 1 & \bar{y}_2 \\ 1 & \bar{y}_3 \\ \cdot & \cdot \\ 1 & \bar{y}_{n_{st}} \end{bmatrix} \cdot \begin{bmatrix} \frac{1}{A} & 0 \\ 0 & -\frac{1}{I} \end{bmatrix} \cdot \begin{bmatrix} F_{x,B} \\ M_{z,B} \end{bmatrix} \tag{2}$$

$$\underline{p}_{x,T} = \underline{X}_y \cdot \underline{I}_y \cdot \underline{F}_{x,S} \qquad \underline{X} = \underline{X}_y \cdot \underline{I}_y \tag{2a}$$

where *A* and *I* are the area and the moment of inertia of the beam section, respectively. A similar equation can be written for the shear force and the forces in y-direction.

Step 2: Determination of nodal forces in the target system for the stress pattern according to step 1

The stresses are now applied as “distributed loads” to the elements of the target system. In the case of the beam-plate-problem the nodal forces corresponding to these element stresses are for one element

$$\begin{bmatrix} F_{x1} \\ F_{x2} \end{bmatrix} = t \cdot \int \begin{bmatrix} \frac{1}{2} - \frac{\bar{y}}{a} \\ \\ \frac{1}{2} + \frac{\bar{y}}{a} \end{bmatrix} \cdot p_x \cdot d\bar{y} \tag{3}$$

$$\underline{F}_{x,T}^{(el)} = t \cdot \int \underline{N} \cdot p_x \cdot d\bar{y} \tag{3a}$$

Here, $\underline{N}$ denotes the Matrix with the interpolation functions, p_x describes the distributed loads and *t* is the thickness of the plate (see Fig. 2).

The beam-plane stress element problem is formulated for finite elements with linear shape functions $\underline{N}$. The same functions can be used to interpolate the stresses between the nodal points as

$$p_x = \underline{N}^T \cdot \underline{p}_x^{(el)} \tag{4}$$

with

$$p_x^{(el)} = \begin{bmatrix} p_{x1} \\ p_{x2} \end{bmatrix} \quad (4a)$$

Introducing (4) into (3a) gives

$$\underline{F}_x^{(el)} = t \cdot \int \underline{N} \cdot \underline{N}^T \cdot d\bar{y} \cdot \underline{p}_x^{(el)} \quad (5)$$

or by integration

$$\underline{F}_x^{(el)} = \underline{A}_x^{(el)} \cdot \underline{p}_x^{(el)} \quad (6)$$

with $$\underline{A}_x^{(el)} = \frac{a \cdot t}{6} \cdot \begin{bmatrix} 2 & 1 \\ 1 & 2 \end{bmatrix}. \quad (7)$$

Equation (6) gives the nodal forces for a single element of the target system. The relationship is now extended to all elements of the target system which are connected with the source system as

$$\underline{F}_{x,T} = \underline{A}_x \cdot \underline{p}_{x,T} \quad (8)$$

The Matrix $\underline{A}_x$ is obtained by assembling the element matrices $\underline{A}_x^{(el)}$ of all elements connected to the source system. The element assemblage procedure is the same as for element stiffness matrices. The general relationship corresponding to (8) is written as

$$\underline{F}_T = \underline{A} \cdot \underline{p}_T \quad (9)$$

general, the matrix A depends on the finite element type in the target system as well as on the In function describing the stress variation in the source system. For quadrilateral plate elements in bending the matrix $\underline{A}$ is given in [1].

Step 3: Transformation of the stiffness matrix of the source system

The transformation matrix for the element forces can be obtained easily with (1) and (9) as

$$\underline{F}_T = \underline{T}^T \cdot \underline{F}_S \quad (10)$$

where

$$\underline{T}^T = \underline{A} \cdot \underline{X}\ . \quad (11)$$

The node displacements are $\underline{u}_S$ and $\underline{u}_T$ in the source and in the target system, respectively. It can be shown that they are also transformed with the matrix $\underline{T}$ as

$$\underline{u}_S = \underline{T} \cdot \underline{u}_T \quad (12)$$

In the source system the stiffness matrix is given by

$$\underline{K}_S \cdot \underline{u}_S = \underline{F}_S \quad (13)$$

Using (10) and (12) it can be transformed into the target system by

$$\underline{K}_T \cdot \underline{u}_T = \underline{F}_T \quad (14)$$

with

$$\underline{K}_T = \underline{T}^T \cdot \underline{K}_S \cdot \underline{T} \quad (15)$$

3.2 EST for a beam element and plane stress elements

The application of the procedure described above is shown for the connection of a beam element with isoparametric plane stress elements with linear interpolation functions. In addition to the normal force and the bending moment according to (8), the shear force has to be transformed. The shear force results in forces in y-direction in the target system.

The beam representing the source system has the degrees of freedom and nodal forces (see Fig. 2(b)) to be connected with the plate

$$\underline{u}_S = \underline{u}_a = \begin{bmatrix} u_B \\ v_B \\ \varphi_B \end{bmatrix} \qquad \underline{F}_S = \underline{F}_a = \begin{bmatrix} F_{x,B} \\ F_{y,B} \\ M_{z,B} \end{bmatrix} \quad . \tag{16}$$

The stiffness matrix of the beam is given by

$$\begin{bmatrix} \underline{K}_{aa} & \underline{K}_{ab} \\ \underline{K}_{ba} & \underline{K}_{bb} \end{bmatrix} \cdot \begin{bmatrix} \underline{u}_a \\ \underline{u}_b \end{bmatrix} = \begin{bmatrix} \underline{F}_a \\ \underline{F}_b \end{bmatrix} \tag{17}$$

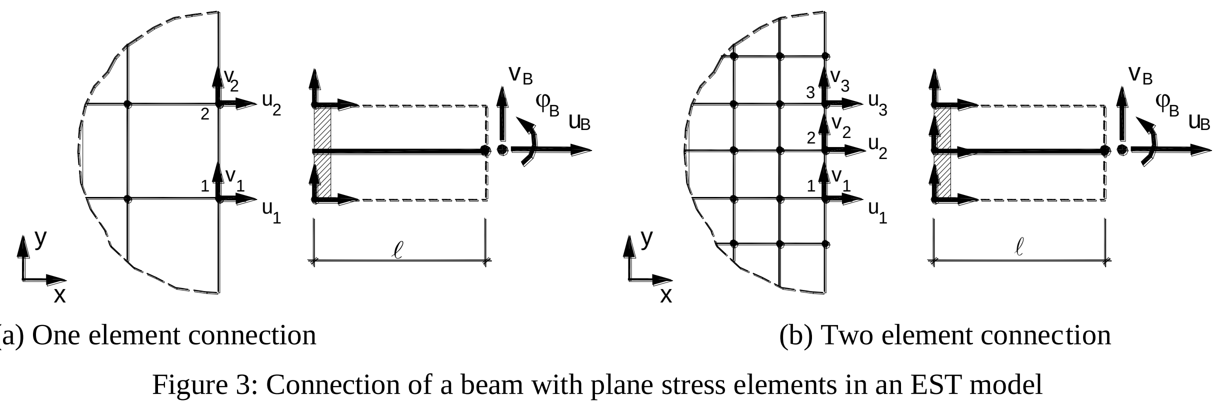
with $\underline{K}_{ba} = \underline{K}_{ab}^T$ and

$$\underline{K}_{aa} = \begin{bmatrix} k_{uu} & 0 & 0 \\ 0 & k_{vv} & k_{v\varphi} \\ 0 & k_{v\varphi} & k_{\varphi\varphi} \end{bmatrix}, \quad \underline{K}_{ab} = \begin{bmatrix} -k_{uu} & 0 & 0 \\ 0 & -k_{vv} & k_{v\varphi} \\ 0 & -k_{v\varphi} & k_{\varphi\varphi}/2 \end{bmatrix}, \quad \underline{K}_{bb} = \begin{bmatrix} k_{uu} & 0 & 0 \\ 0 & k_{vv} & -k_{v\varphi} \\ 0 & -k_{v\varphi} & k_{\varphi\varphi} \end{bmatrix}$$

where $k_{uu} = \frac{E \cdot A}{\ell}$, $k_{vv} = 12 \cdot \frac{E \cdot I}{\ell^3}$, $k_{\varphi v} = 6 \cdot \frac{E \cdot I}{\ell^2}$, $k_{\varphi\varphi} = 4 \cdot \frac{E \cdot I}{\ell}$.

Two meshes of the target system are considered, see Fig. 3. The procedure according to section 3.1 gives if one element with linear interpolation function connected with the beam element (see Fig. 3(a))

$$\underline{T} = \begin{bmatrix} 1/2 & 0 & 1/2 & 0 \\ 0 & 1/2 & 0 & 1/2 \\ 1/d & 0 & -1/d & 0 \end{bmatrix} \qquad \underline{u}_T = \begin{bmatrix} u_1 \\ v_1 \\ u_2 \\ v_2 \end{bmatrix}. \tag{18}$$



(a) One element connection (b) Two element connection

Figure 3: Connection of a beam with plane stress elements in an EST model

For a connection with two elements (Fig. 3(b)) the transformation matrix is

$$\underline{T} = \begin{bmatrix} 1/4 & 0 & 1/2 & 0 & 1/4 & 0 \\ 0 & 1/8 & 0 & 3/4 & 0 & 1/8 \\ 1/d & 0 & 0 & 0 & -1/d & 0 \end{bmatrix} \qquad \underline{u}_T = \begin{bmatrix} u_1 \\ v_1 \\ u_2 \\ v_2 \\ u_3 \\ v_3 \end{bmatrix}. \tag{18a}$$

The application of equation (12) $\underline{u}_S = \underline{T} \cdot \underline{u}_T$ illustrates the averaging process of the transformation matrix for the displacements.

The transformation of node *a* by EST gives the modified stiffness matrix of the beam

$$\begin{bmatrix} \underline{T}^T \cdot \underline{K}_{aa} \cdot \underline{T} & \underline{T}^T \cdot \underline{K}_{ab} \\ \underline{K}_{ba} \cdot \underline{T} & \underline{K}_{bb} \end{bmatrix} \cdot \begin{bmatrix} \underline{u}_T \\ \underline{u}_b \end{bmatrix} = \begin{bmatrix} \underline{F}_T \\ \underline{F}_b \end{bmatrix} \tag{19}$$

Node *b* can be transformed similarly, if necessary.

4 Applications

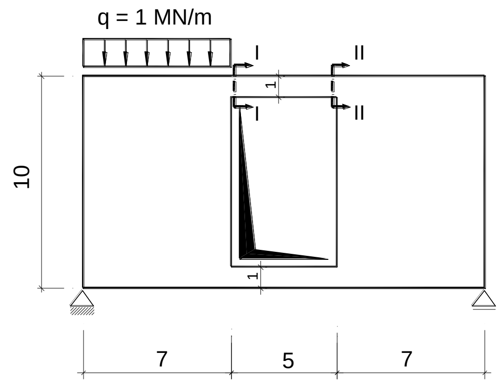
4.1 Deep beams

A deep beam with a large opening is taken as example, Fig. 4 [2]. The thickness of the wall is 0,5 m, modulus of elasticity 30000 kN/m^2 and Poisson ration 0. Two models are analyzed. In the first model only plane stress elements are used. In the second model the beam-type domain of the wall above and below the opening is modeled by beam elements. In this model beam and plane stress elements are connected by EST according to section 3.2.

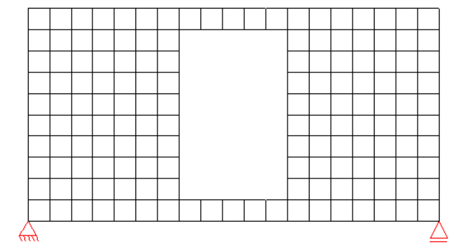
The stress resultants in the sections I-I and II-II of the beam-type domain above the opening are given in Table 1. In model 1 they are evaluated by integration of the corresponding plate element stresses whereas in model 2 they are obtained directly as sectional forces of the beam element. The integration in model 1 is based on element stresses instead of nodal stresses because of the discontinuity of stresses in the sections considered. The results demonstrate the accuracy of the EST beam element. With only one plate element connected with the beam element (i.e. e = 1 m) sectional forces with high accuracy are obtained. Model 1 converges to this solution for smaller mesh size only.

Table 1: Sectional forces.

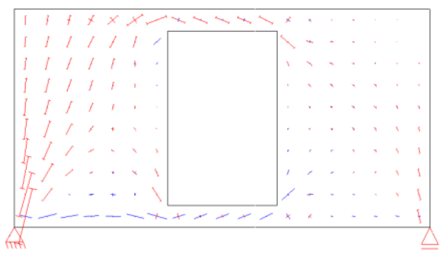
Section	Sectional force	Model 1, e = 1.0 m	Model 1, e = 0.5 m	Model 2, e = 1.0 m
I-I	Normal force [kN]	1364	1364	-1362
	Bending moment [kNm]	837	1553	1592
	Shear force [kN]	-642	-642	-643
II-II	Normal force [kN]	-1364	-1364	-1362
	Bending moment [kNm]	875	1632	1621
	Shear force [kN]	-642	-642	-643



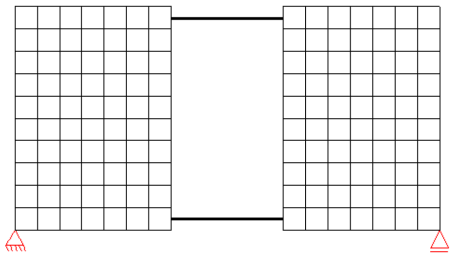
(a) System



(b) FE model 1: plate elements, e= 1.0 m



(c) Principal stresses, FE model 1

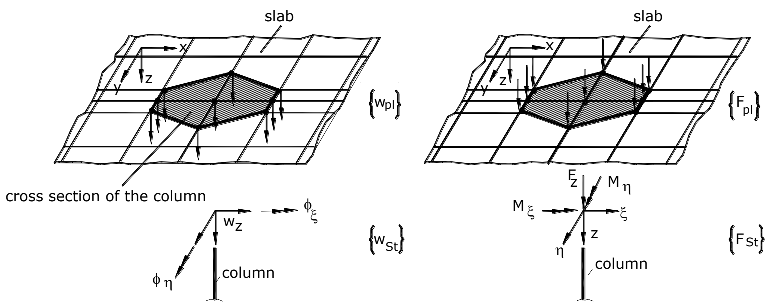


(d) FE model 2: plate and beam elements, e= 1.0 m

Figure 4: Deep beam with a large opening

4.2 Flat slabs

In the analysis of concrete flat slabs the connection of the column and the slab requires special models. A consistent approach is the obtained by EST. The stiffness matrix of the column which is modeled as beam is transformed by EST to the finite element system of the plate in bending, representing the slab. The transformation matrix for the columns-slab connection and applications are given in [1], [3], [4], Fig. 5.



Nodal displacements

Nodal forces

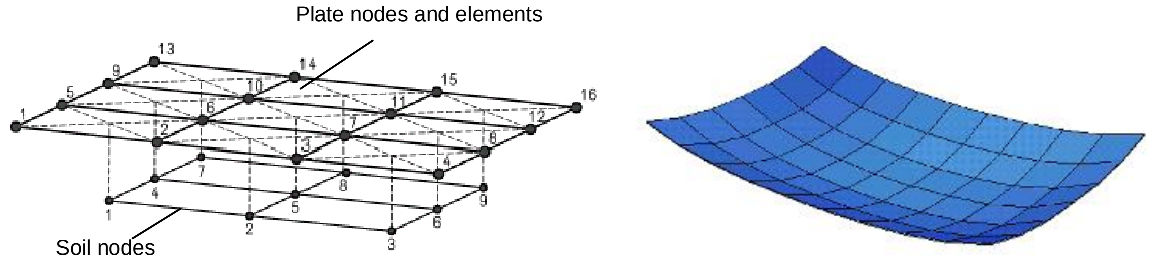
Figure 5: Column-slab modeling by EST

4.3 Foundation slabs

In the analysis of foundation slabs the interaction of the soil and the plate in bending is represented by a stiffness matrix of the soil. Assuming that the soil pressure acting on a plate element is uniformly distributed the displacement w at the surface of a homogeneous elastic halfspace is obtained as

$$w(x,y) = \frac{1-\mu^2}{\pi \cdot E} \cdot \iint_A \frac{p(\varsigma,\eta)}{(x-\varsigma)^2 + (y-\eta)^2}\, d\varsigma\, d\eta\,, \tag{20}$$

where E denotes the modulus of elasticity and μ the Poisson ratio of the soil. The integration has to be done over the plate element area. Using this relationship the flexibility matrix of the soil can be to be computed. Its inverse is the stiffness matrix of the soil. It relates to the midpoints of the plate elements for the foundation slab (see Fig. 5). The transformation to the nodes of the foundation slab can be done by EST. For rigid foundations the results agree well with analytical solutions [5].



(a) FE-model of the slab and nodes for Soil-Structure-Interaction (b) Deformation of a foundation slab

Figure 5: Foundation slab

5 Conclusions

The EST is well suited to model the connection between domains of finite elements with different stress systems. The results agree well with more sophisticated finite element models. EST can be applied to use structural elements with different stress systems as beams with plate elements in the same finite element model. In EST-beams the sectional forces which are required in RC design, are obtained directly without integration of element stresses. EST elements can be developed for many dissimilar structural elements which can be assembled the same finite element model.

References

[1] H. Werkle, *Finite Elemente in der Baustatik*, 2nd Edition, Vieweg, Wiesbaden, Germany (2001).

[2] M. Schneider, *Finite-Element-Berechnung von Scheibentragwerken*, Diplomarbeit Fach-hochschule Konstanz, Fachbereich Bauingenieurwesen (2001)

[3] H. Werkle, *Konsistente Modellierung von Stützen bei der Finite-Element-Berechnung von Flachdecken*, Bautechnik, 77 (2000), Ernst&Sohn, Berlin, Germany, 416-425.

[4] H. Werkle, *Analysis of flat slabs by the finite element method*, ASEM'02, Pusan, Korea (2002)

[5] G. Nerz, M. Herr, *Vergleichende Untersuchungen zur Boden-Bauwerk-Wechselwirkung mit der Methode der Finiten Elemente*, Diplomarbeit Fachhochschule Konstanz, Fachbereich Bauingenieurwesen (2001)