A Transformational Approach to Teaching Matrix Structural Analysis, and Visual Implementation using *Mathcad*

Author:

Finley A. Charney, Virginia Tech, Blacksburg VA, fcharney@vt.edu

INTRODUCTION

At most universities, matrix methods of structural analysis are taught in the senior year, or in the first year of graduate study. For students taking the course, the material may be very challenging because it has been several semesters since they have taken the prerequisite courses (generally linear algebra and theory of structures). More problematically the course is challenging because many of the students have had little exposure to computer programming.

At Virginia Tech, the course Computer Methods of Structural Analysis I (CEE 4404) has been designed to minimize these challenges, but still provide a firm theoretical basis in matrix structural analysis. The theoretical basis of the course is rooted in the concepts of equilibrium, compatibility, and superposition (requiring linear-elastic constitutive laws), and is presented in the context of four different levels of "Scope" within a structure. A key aspect of the course is a heavy reliance on a variety of mathematical transformations that relate the levels of scope to each other. Because of the reliance on transformations, the methodology described in this paper is termed the "Transformational Approach" to teaching matrix structural analysis.

The implementation of the method is facilitated through the use of the commercial mathematics program *Mathcad*[1]. *Mathcad* is used in two ways; first as a visual matrix manipulation tool, and second, as a framework for writing complete structural analysis programs. While a variety of programming platforms could be used (e.g. C++, C#, Visual Basic, *Matlab*[2], *Mathematica*[3]) *Mathcad* was chosen because it is highly visual, relatively easy to learn, and is widely used in the structural engineering profession.

By the end of the semester, quite complex problems may be solved with *Mathcad*, including any two-dimensional structure incorporating frame or truss elements. Practically any type of loading may be considered; shear deformations, rigid ends, and member end-releases may be included; and a variety of constraints may be modeled. Aside from *Mathcad*, no commercial structural analysis software is used in the course.

THEORETICAL BASIS

The theoretical material in the course is taught from the perspective of mathematical transformations. These transformations are presented on the first day of class in the context of the basic equilibrium equations (1a), and an expanded version of the same equations (1b).

$$K\Delta = F \tag{1a}$$

---

[1] Product Development Company , Needham, MA.
[2] The MathWorks, Inc., Natick, MA.
[3] Wolfram Research, Champaign, IL.

$$\left[\sum_{i=1}^{nels} B_i U_i b_i \tilde{k}_i a_i T_i A_i\right]\Delta = P - \sum_{i=1}^{nels} B_i U_i \hat{q}_i \tag{1b}$$

In (1a), K is the system stiffness matrix, F is a vector of nodal loads, and Δ is a vector containing the displacements at the various degrees of freedom (DOF). In the expanded form, the matrix product in the summation on the left hand side represents a progressive change of scope, starting with the most fundamental "reduced" element stiffness $\tilde{k}$, and ending with the same element ready for assembly into the system's global stiffness matrix. This expansion is facilitated by displacement transformations a, T, and A, and by corresponding force transformations b, U, and B. Similarly, the matrix product in the summation on the right hand side expands the element fixed-end forces $\hat{q}$, enabling them to be added into the global force vector. The limit *nels* in the summations of (1b) is the number of elements in the structure.

While (1b) is initially intimidating to most students, it may be easily explained through a description of the various transformations that are involved. Each transformation relates two adjacent levels of scope. For any element of any structure, there are only four levels of scope, as indicated in Table 1, and as illustrated in Figure 1. Each level of scope consists of two parts; the chosen degree-of-freedom set, and the reference coordinate system.

Level of Scope	DOF Set	Reference Coordinate System	Diacritical Identifier
1 (narrowest)	Reduced Element	Local	Tilde (~)
2	Full Element	Local	Carat (^)
3	Full Element	Global	Overbar (-)
4 (broadest)	Structure	Global	none

TABLE 1 - Levels of Scope used in Analysis of Any Structure

At Level 1, a reduced element DOF set is used. The physical model for this level of scope is presented in Figure 1a for an element of a planar frame. All variables referring to this element are presented with a tilde (~) diacritical mark. The term "reduced" indicates that a partial set of element DOF are utilized. The basic requirements in choosing the reduced DOF set are that the element be stable and statically determinate. For most elements there is some choice in the selection of the active DOF set in the reduced element [McGuire, et al., 2000].

The first step in the Level 1 analysis is to develop the flexibility matrix $\tilde{d}$ of the element. Each column of $\tilde{d}$ is found by applying a unit force at DOF j and computing the resulting displacements $\tilde{d}_{i,j}$ at each degree of freedom i. Once the flexibility matrix is found, the principle of superposition is invoked to show that any displacement pattern can be represented as a linear combination of the displacements arising from the unit forces:

$$\tilde{\delta} = \tilde{d}\,\tilde{f} \tag{2}$$

where $\tilde{\delta}$ is the nodal displacement vector resulting from applied nodal forces $\tilde{f}$. If the individual terms in $\tilde{d}$ are computed using virtual work, it is easy to show that $\tilde{d}$ must be symmetric, because the only difference between $\tilde{d}_{i,j}$ and $\tilde{d}_{j,i}$ is that the real and virtual loadings are switched when computing these terms.

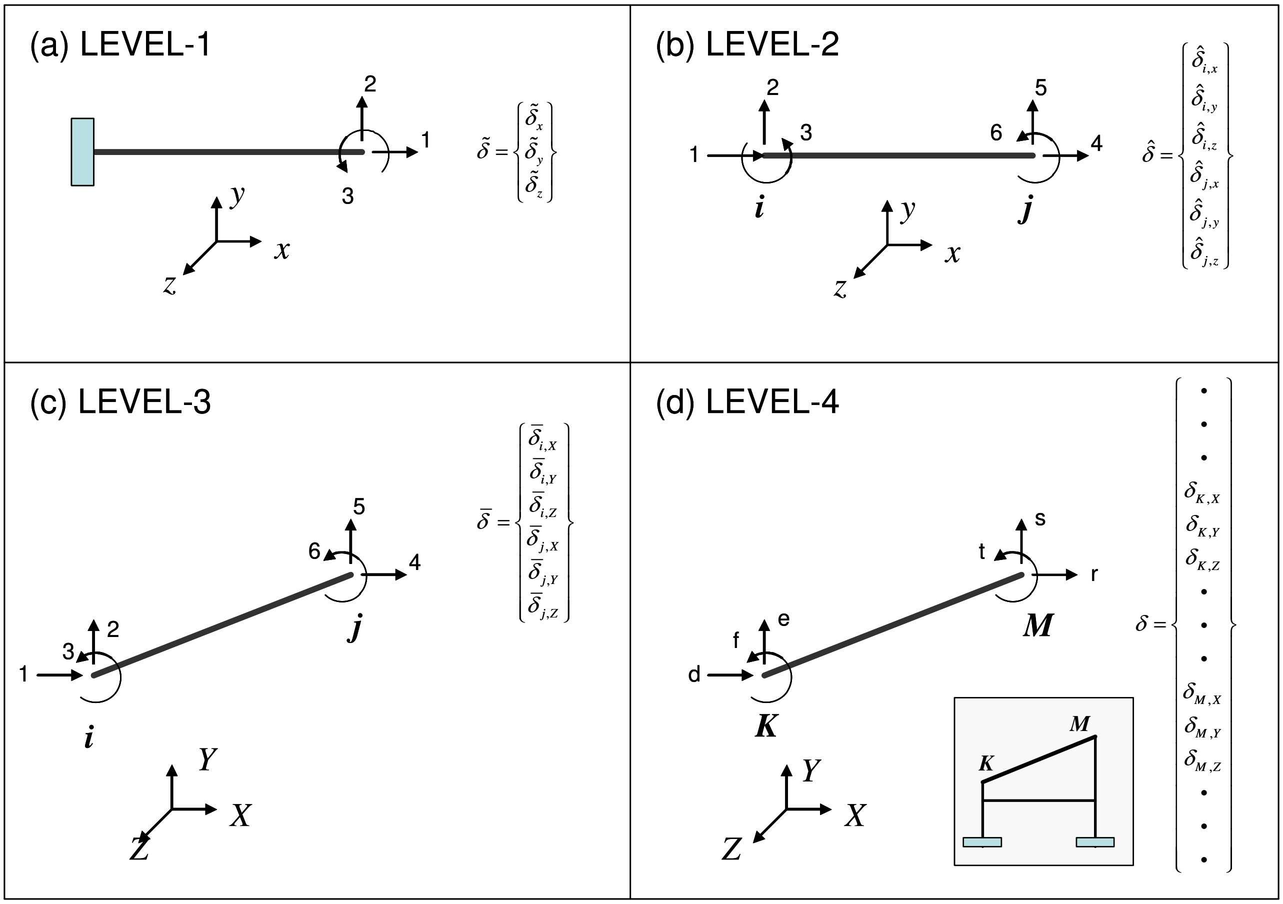

FIGURE 1 - Levels of Scope for 2-D Frame Element

If both sides of (2) are premultiplied by the $\tilde{d}^{-1} = \tilde{k}$, the equilibrium equations for the reduced element are formed:

$$\tilde{k}\tilde{\delta} = \tilde{f} \tag{3}$$

Before moving to the next level of scope, it is important to make a few points about the work done at Level 1:

- Physically, the displacements $\tilde{\delta}$ in the element represent the *deformations* in the element.
- The flexibility $\tilde{d}$ will also be used for the formulation of element fixed end forces
- Direct formulation of $\tilde{k}$ is possible by imposing unit displacements at the various DOF. In some cases, however, a statically indeterminate analysis is required. This difficulty is avoided by forming $\tilde{d}$ and inverting to form $\tilde{k}$ (in which case the indeterminate nature of the problem is handled "automatically" through the inversion process).