

FINITE ELEMENT STRUCTURAL ANALYSIS ON AN EXCEL SPREADSHEET

INTRODUCTION

Finite Element (FE) software is an essential tool for most structural design engineers, and at the cost of most commercial FE software, it had better be essential. The commercial FE software used by many engineering firms will provide you with more computer-output than you could read in a month and more than you can understand in a year. Commercial programs are great for impressing clients, and great for performing extensive analysis when really needed. But in design of frame-type structures, rarely is all that power and output really needed.

Spreadsheet based FE analysis differs from conventional FE analysis as follows:

Spreadsheet-based Finite Element	Conventional Finite Element
Data, equations, results on one spreadsheet	Huge amounts of data & equations
Formulas are all on one spreadsheet	Complex “black box” algorithms
Calculation steps and intermediate calculation results all on one spreadsheet	Complex theory, beyond average user
Spreadsheets are best for handling huge amounts of data and equations	Commercial software is best for handling complex algorithms and complex theory
The FE spreadsheet is free and most engineers already have the software necessary for spreadsheets.	Commercial software is expensive but needed.

Understanding FE theory allows the user to in many cases forego commercial software and use more basic software, such as the FE spreadsheet. In 10 years of private practice, I have relied almost exclusively on a FE spreadsheet for analyzing frame-type structures. That spreadsheet is presented in this course as a teaching tool and as a practical, effective design tool. The spreadsheet is limited to 2-dimensional frames of about 50 nodes, but if a problem is within that range it is easier to use, easier to understand, easier to port, easier to check and much less expensive than commercial programs.

In 25 years of engineering, I have never seen a design that was flawed because the designer failed to generate enough computer output. I have never seen a structure that was inadequate because the designer didn't use enough nodes in his analysis model. I have never seen an analysis that was erroneous because there weren't enough digits to the right of the decimal point. For most frame-type structure problems, use of commercial FE software results in too much output, too many nodes, and too many insignificant digits.

When you buy a new car, you go for a drive before you read the owners manual. In a similar fashion, in this course you will download and use the FE spreadsheet before you learn the underlying theory. Since theory is easier to understand as an explanation of a tool we can use than as a hypothesis of one we could create, we will first learn how to use the FE spreadsheet, then use theory to explain how it works.

Section 1: Description of the Finite Element Spreadsheet:

Two spreadsheet workbooks in Microsoft Excel format are provided for download as a part of this course. They are both FE spreadsheets; one is a training sheet with just 5 nodes and 5 members, the other is a sheet for practical use with 16 nodes and 37 members. Each workbook consists of three sheets:

1. a documentation sheet,
2. a FE analysis sheet,
3. a plot sheet.

1. The documentation sheet gives an overview of the structure of the FE spreadsheet and a list of the basic underlying assumptions.

2. The FE analysis sheet provides all the formulas and calculations to solve frame-type 2D static problem. Required inputs are:

- node coordinates,
- member node-to-node connectivity,
- member properties,
- support conditions,
- and loadings.

Calculated output (on the same sheet) is:

- support reactions,
- node displacements,
- member end forces,
- all intermediate calculations.

3. The plot sheet that shows node and member geometry to assist in verifying model input.

Section 2: Definitions ~ Element Properties:

(these definitions are in the context of the FE spreadsheet provided with this course, and may have other or broader meaning in other contexts)

- A frame member has both axial and flexural stiffness properties.
- A beam-type member is a frame member with axial stiffness approaching zero.
- A truss-type member is a frame member with flexural stiffness approaching zero.
- In the FE spreadsheet provided with this course, frame members (including special beam-type and truss-type) are the only elements allowed. (Plate-type elements and shear-strain elements are not allowed).
- A frame is a structure composed of any number of frame-type members, joined at nodes, with axial and flexural stiffness continuity at the nodes.
- “Axx” is member cross-sectional area perpendicular to the local x-axis.
- “Izz” is member moment-of-inertia about the local z-axis.
- “E_mod” is material modulus of elasticity.

Section 3: Definitions ~ Local and Global Coordinates:

- The FE spreadsheet is for 2D members in the x-y plane.
- Local coordinates are relative to the member; global coordinates are consistent for all members. There are as many local coordinate systems as there are members, but there is only one global coordinate system.

- Local x-axis is along the member axis, with positive being from the “i”-end toward the “j”-end.
- Right hand rule applies such that if the x-axis points East and the y-axis North, the z-axis points up. Similarly, if the x-axis points East and the y-axis points up, then the z-axis points South.) Right hand rule is that if you point your thumb up, your pointer finger straight ahead and your index finger at right angles to both (toward your left hand), your pointer defines the x-axis, your index defines the y-axis and your thumb defines the z-axis. If you use this rule to remember the axis orientations it is a good idea to write x, y & z on the respective digits because if you transpose letters to the wrong digits your axes can also be transformed to left-hand orientation.
- Typically, lower case letters represent local coordinates, capital letters represent global coordinates. In matrix notation, lower case letters indicate local member properties, with respect to local axis and upper case represent global properties.

Section 4: Simple Beam Example on the FE Spreadsheet

Figures 1A & 1B show a simple beam problem, including a sketch of the model, the input loads and the resulting forces and deflections. To mirror the results:

- download the spreadsheet FE 5N 5M.xls (for Finite Element 5 Node 5 Member)
- save the original
- save a working copy as FE_Sec4.xls
- change the input cells (colored yellow) to match Figure 1A
- verify that calculated results match Figure 1A

The structure model and input are annotated in Figure 1A. On this spreadsheet, the number of nodes is set at 5 and the number of members is set at 5. All nodes must be connected and all members must be used. If a problem requires more nodes or more members a larger version of the spreadsheet is required. If a problem requires fewer than 5 nodes or members this spreadsheet may be used, with the extraneous nodes and members inactivated. This is in contrast to typical commercial FE programs, in which the user selects the number of nodes and members.

To inactivate a member, set properties A_{xx} and I_{zz} to near zero. In some cases the values can be set to zero, but in most cases setting properties to zero will result in a spreadsheet “!NUM” error message. To inactivate a node, connect to it only with inactivated members. The Figure 1 problem requires 5 nodes but only requires 4 members so member 5 is inactivated by setting its properties to near zero.

Calculated results are annotated in Figure 1B. Be careful of sign convention with respect to output. Coordinate axes are per right hand rule (per the Definitions section previously and as shown in Fig 1B.), and results follow accordingly. Note also that consistent units must be used.

The Figure 1 Example has input of node-point loads of 2.0, 3.0 and 4.0 at nodes 2, 3 & 4 respectively. The resulting maximum moment is 60 at member 3, end “i”, maximum deflection is 0.95 at node 3.

FIGURE 1A BEAM EXAMPLE ~ WITH MODEL SKETCH AND INPUT ANNOTATION

5 NODE, 5 MEMBER SPREADSHEET

Rev: 5/29/08

Use high spring values for fixed supports

Yellow cells accept input.

[len]			[for/len]	[for/len]	[for/len]	[for*len]	[force]	[force]	[for*len]	[force]	[force]	[rad]	[length]	[length]
NODE DATA:			Support Springs			Input Forces			Support Reactions			Output Deflections		
Node	x	y	k_rot	k_y	k_x	Mom	FY	FX	Mom	FY	FX	Rot	Dy	Dx
1	0.0	0.0		999999	999999				0	-4	0	0.073	0.000	0.000
2	10.0	0.0					2.00		0	0	0	0.053	0.658	0.000
3	20.0	0.0					3.00		0	0	0	0.003	0.950	0.000
4	30.0	0.0					4.00		0	0	0	-0.052	0.692	0.000
5	40.0	0.0		999999					0	-5	0	-0.077	0.000	0.000

			[len^2]	[len^4]	[for/len^2]	[for/len]				[for*len]	[force]	[force]	[for*len]	[force]	[force]
MEMBER DATA:								Output Member Forces							
Mem	i_node	j_node	Axx	Izz	E_mod	Uni_Load	Len		Mi	Vi	Pxi	Mj	Vj	Pxj	
1	1	2	0.001	10	1000		10.00		0	-4	0	-40	4	0	
2	2	3	0.001	10	1000		10.00		40	-2	0	-60	2	0	
3	3	4	0.001	10	1000		10.00		60	1	0	-50	-1	0	
4	4	5	0.001	10	1000		10.00		50	5	0	0	-5	0	
5	5	1	0.001	0.001	0.001		40.00		0	0	0	0	0	0	

Number of nodes & members is fixed. Each/all must be included in the model.

Inactivate mem 5 by setting properties near zero (but $\neq 0$.)

Input Loads

calculated output results.

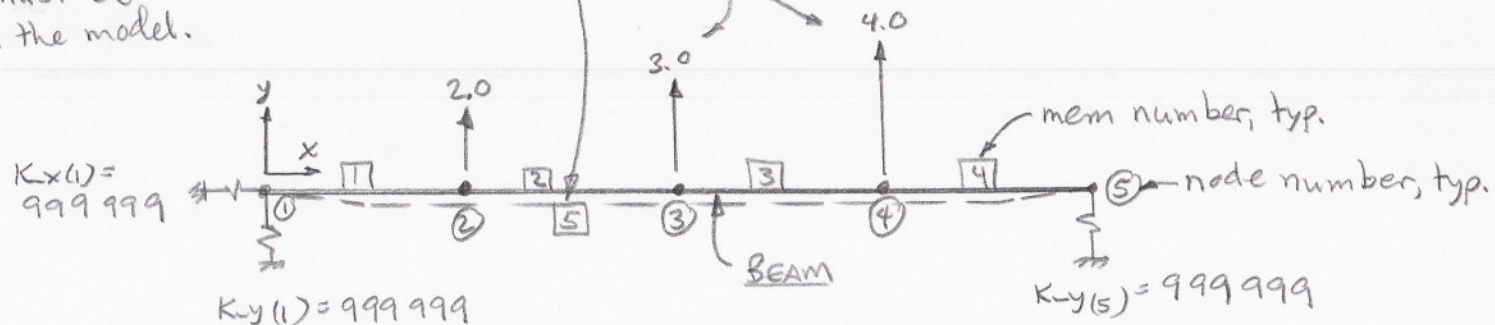


FIGURE 1A: BEAM MODEL w/ INPUT ANNOTATION

FIGURE 1B BEAM EXAMPLE ~ WITH OUTPUT ANNOTATION
5 NODE, 5 MEMBER SPREADSHEET

Rev: 5/29/08

[len]			[len]	[for/len]	[for/len]	[for/len]	[for*len]	[force]	[force]	[for*len]	[force]	[force]	[rad]	[length]	[length]
NODE DATA:			Support Springs			Input Forces			Support Reactions			Output Deflections			
Node	x	y	k_rot	k_y	k_x	Mom	FY	FX	Mom	FY	FX	Rot	Dy	Dx	
1	0.0	0.0		999999	999999				0	-4	0	0.073	0.000	0.000	
2	10.0	0.0					2.00		0	0	0	0.053	0.658	0.000	
3	20.0	0.0					3.00		0	0	0	0.003	0.950	0.000	
4	30.0	0.0					4.00		0	0	0	-0.052	0.692	0.000	
5	40.0	0.0		999999					0	-5	0	-0.077	0.000	0.000	

MEMBER DATA:			[len^2]	[len^4]	[for/len^2]	[for/len]	[for*len]	[force]	[force]	[for*len]	[force]	[force]	
Mem	i_node	j_node	Axx	Izz	E_mod	Uni_Load	Len	Output Member Forces					
								Mi	Vi	Pxi	Mj	Vj	Pxj
1	1	2	0.001	10	1000		10.00	0	-4	0	-40	4	0
2	2	3	0.001	10	1000		10.00	40	-2	0	-60	2	0
3	3	4	0.001	10	1000		10.00	60	1	0	-50	-1	0
4	4	5	0.001	10	1000		10.00	50	5	0	0	-5	0
5	5	1	0.001	0.001	0.001		40.00	0	0	0	0	0	0

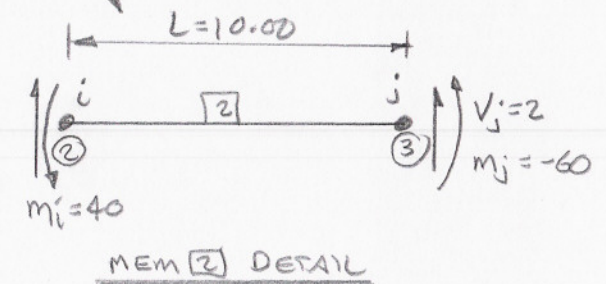
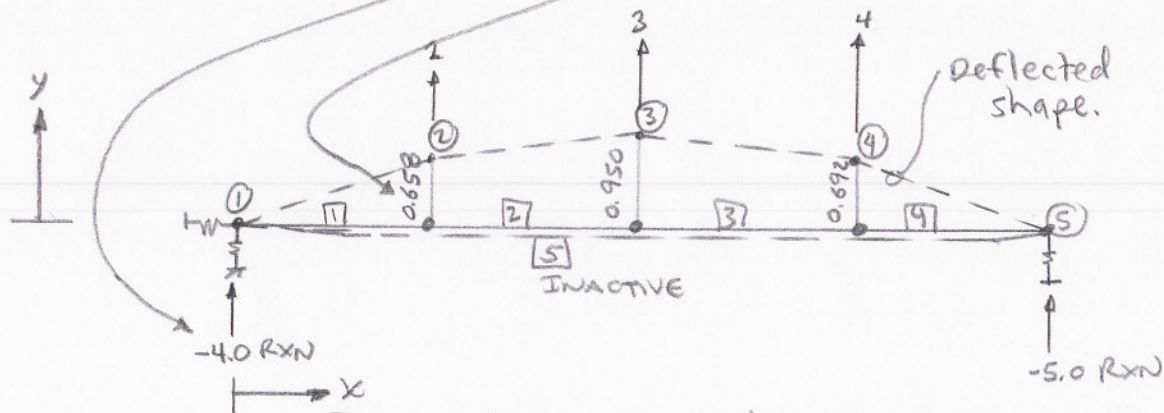


FIGURE 1B: MODEL w/ OUTPUT ANNOTATION

Section 5: Truss Example on the FE Spreadsheet ~ [Ref: Figure 2]

Figure 2 Example is a triangular truss, with a vertical 3.0 load and a horizontal 2.0 load, both at node 3.

A truss is essentially a frame with no flexural resistance. Therefore, to analyze a truss the member moment of inertia needs to be set near zero (it cannot be set to zero or an Excel error message “#NUM!” will result). To mirror Figure 2 results, copy the input from Fig. 2 to your spreadsheet. Verify that your output matches Fig 2.

Note that each node is at the end of at least one member (no nodes are unconnected). To see the effect of having an unconnected node, change member 5 connectivity from 5-2 to 3-2 (by changing cell B21 from “5” to “3”) such that node 5 is not connected to any members. Output values all change to “!NUM” error message.

Note that node 1, 4 & 5 can have the same coordinates (0,0), but that you can't connect a member between two identical nodes because the member will have zero length. To see the effect of a zero length member, change node 4 ordinates from (0,0) to (10,0) such that it has the same coordinates as node 2. Output values all change to “#DIV/0!” error message.

The result summary for the Figure 2 truss example is that node 3 has calculated deflections of 0.003 vertical and 0.002 horizontal. The maximum member axial force is 3.54 in member 3.

FIGURE 2 TRIANGULAR TRUSS EXAMPLE

5 NODE, 5 MEMBER SPREADSHEET

Rev:

5/29/08

[len]			[for/len]			[for*len]			[force]			[for*len]			[force]			[rad]			[length]		
NODE DATA:			Support Springs			Input Forces			Support Reactions			Output Deflections											
Node	x	y	k_rot	k_y	k_x	Mom	FY	FX	Mom	FY	FX	Rot	Dy	Dx									
1	0.0	0.0		999999	999999				0.00	-2.50	-2.00	0.000	0.000	0.000									
2	10.0	0.0		999999					0.00	-0.50	0.00	0.000	0.000	-0.001									
3	5.0	5.0					3.00	2.00	0.00	0.00	0.00	0.000	0.003	0.002									
4	0.0	0.0							0.00	0.00	0.00	0.000	0.005	-0.001									
5	0.0	0.0							0.00	0.00	0.00	0.000	0.005	-0.001									

MEMBER DATA:			[len^2]	[len^4]	[for/len^2]	[for/len]	[for*len]	[force]	[force]	[for*len]	[force]	[force]	
Mem	i_node	j_node	Axx	Izz	E_mod	Uni_Load	Len	Output Member Forces					
								Mi	Vi	Pxi	Mj	Vj	Pxj
1	1	2	7.00	0.001	1000		10.00	0.00	0.00	0.50	0.00	0.00	-0.50
2	2	3	7.00	0.001	1000		7.07	0.00	0.00	-0.71	0.00	0.00	0.71
3	3	1	7.00	0.001	1000		7.07	0.00	0.00	-3.54	0.00	0.00	3.54
4	4	2	0.001	0.001	1000		10.00	0.00	0.00	0.00	0.00	0.00	0.00
5	5	2	0.001	0.001	1000		10.00	0.00	0.00	0.00	0.00	0.00	0.00

MEM [4] & [5]
INACTIVE

For TRUSS, SET $I_{zz} \rightarrow 0$

MEM [4] & [5] INACTIVE

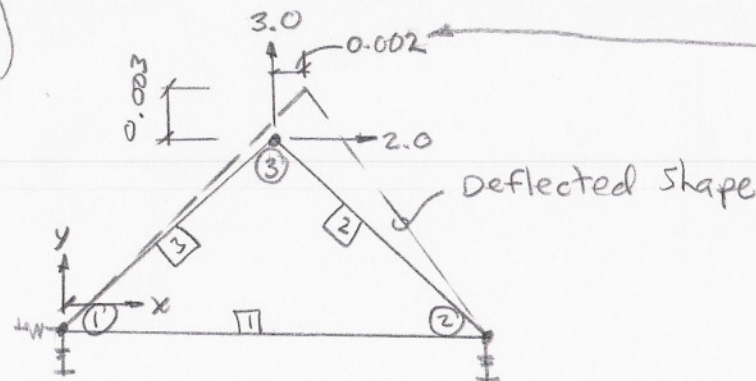


FIGURE 2: TRIANGULAR TRUSS

Section 6: Beam on Elastic Foundation with Uniform Member Load ~ [Ref: Fig 3]

The Figure 3 example is a beam on spring supports. Each support node has a k_y spring value of 80. The beam has varying uniform member loads per Figure 3. To mirror results, copy the input data to your working spreadsheet. Verify that your output matches Fig. 3. Note that Member Data, “Uni_Load” input cells (G17-G21) are for inputting a uniform load perpendicular to the local x-axis of the member (in the local y-direction).

The input/output sections of the FE spreadsheet are divided into two categories: “Node Data” and “Member Data”. Nodes can have supports, applied forces, and deflections. Members have properties, end forces, and can have distributed load. Member end forces are at node locations and are not necessarily maximum values for that member

Resulting deflections for this example vary from -1.0 to -1.8 and maximum moment is 324 at member 2, end “j”. Note that maximums may be greater between nodes. If more detail is required, use more nodes or further analyze critical members as a component problem.

FIGURE 3 BEAM ON ELASTIC FOUNDATION SUBJECT TO UNIFORM MEMBER LOAD

5 NODE, 5 MEMBER SPREADSHEET

Rev:

5/29/08

[len]			[len]	[for/len]	[for/len]	[for/len]	[for*len]	[force]	[force]	[for*len]	[force]	[force]	[rad]	[length]	[length]
NODE DATA:			Support Springs			Input Forces			Support Reactions			Output Deflections			
Node	x	y	k_rot	k_y	k_x	Mom	FY	FX	Mom	FY	FX	Rot	Dy	Dx	
1	0.0	0.0		80	80				0.0	137.0	0.0	0.0	-1.7	0.0	
2	5.0	0.0		80					0.0	140.8	0.0	0.0	-1.8	0.0	
3	10.0	0.0		80					0.0	135.0	0.0	0.0	-1.7	0.0	
4	15.0	0.0		80					0.0	114.7	0.0	0.1	-1.4	0.0	
5	20.0	0.0		80					0.0	82.6	0.0	0.1	-1.0	0.0	

MEMBER DATA:			[len^2]	[len^4]	[for/len^2]	[for/len]	[for*len]	[force]	[force]	[for*len]	[force]	[force]	
Mem	i_node	j_node	Axx	Izz	E_mod	Uni_Load	Len	Output Member Forces					
								Mi	Vi	Pxi	Mj	Vj	Pxj
1	1	2	7.00	50.0	1000	-40	5.00	0	137	0.0	185	63	0.0
2	2	3	7.00	50.0	1000	-20	5.00	-185	78	0.0	324	22	0.0
3	3	4	7.00	50.0	1000	-50	5.00	-324	113	0.0	263	137	0.0
4	4	5	7.00	50.0	1000	-12	5.00	-263	-23	0.0	0	83	0.0
5	5	1	0.001	0.001	1000		20.00	0	0	0.0	0	0	0.0

INACTIVE

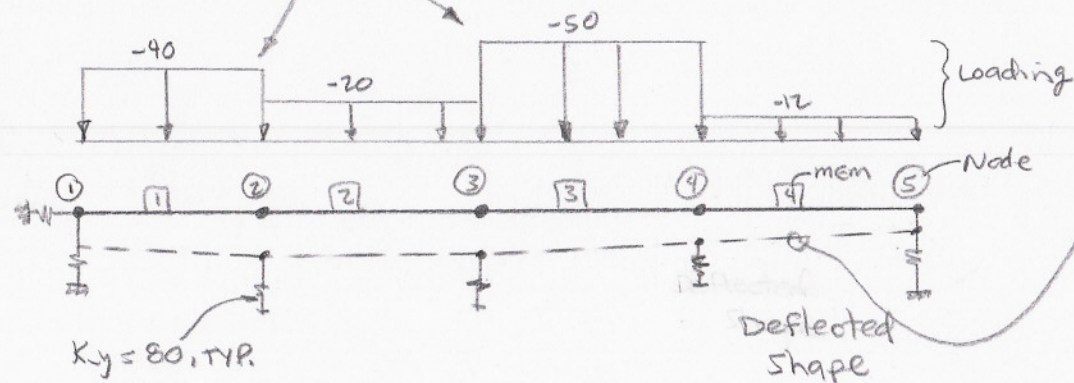


FIGURE 3. BEAM ON ELASTIC SUPPORTS

Section 7: 16-Node Building Example with Member End Release ~ [Ref: Fig 4 & 5]

Previous examples have been limited to a few nodes and a few members to demonstrate the functionality of the spreadsheet. Most design problems require more nodes and members. Figures 4 & 5 show a 16-node, 37-member spreadsheet, which is a functional size for a number of structural design problems. The particular problem shown is a building frame with two 1000 horizontal loads and some -50 uniform member loads. Maximum deflections are -1.7 in the y-direction and 9.9 in the x-direction. Reactions at node 1 are $FY = -1500$ & $FX = -637$. Reactions at node 14 are $FY = 4500$ & $FX = -1363$.

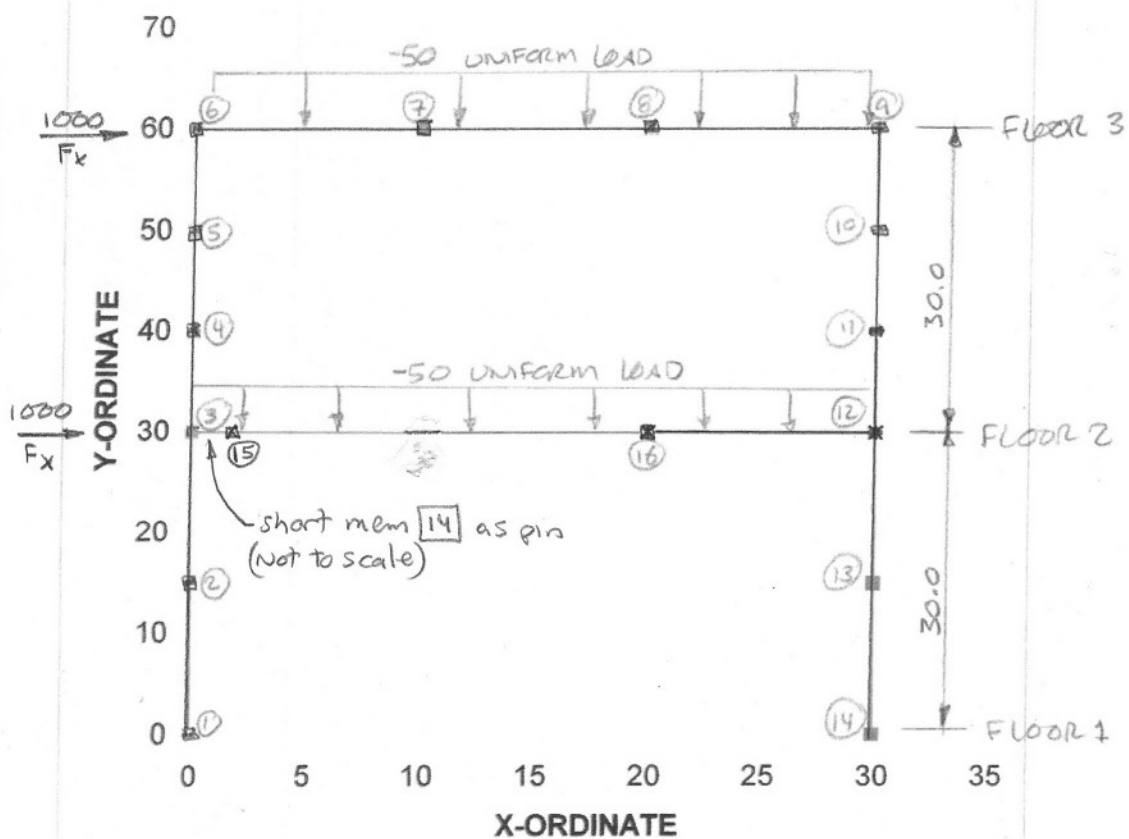
Member end release of member-15 at node-15 is modeled by inserting a very short member-14 with a very low I_{zz} value ($1.0E-08$). Member-14 is essentially a pin of 0.001 diameter and very low friction in this example.

FIGURE 4 3-STORY BUILDING EXAMPLE
16 NODE, 37 MEMBER SPREADSHEET

Rev: 12/12/08

NODE DATA:			Support Springs			Input Forces			Support Reactions			Output Deflections			
Node	x	y	k_rot	k_y	k_x	Mom	FY	FX	Mom	FY	FX	Rot	Dy	Dx	
1	0	0	0	999999	999999				0	-1500	-637.2	-0.22	0.0	0.0	
2	0	15							0	0	0.0	-0.21	0.2	3.2	
3	0	30						1000	0	0	0.0	-0.17	0.4	6.1	
4	0	40							0	0	0.0	-0.14	0.4	7.6	
5	0	50							0	0	0.0	-0.11	0.5	8.9	
6	0	60						1000	0	0	0.0	-0.09	0.5	9.9	
7	10	60							0	0	0.0	-0.07	-0.2	9.8	
8	20	60							0	0	0.0	-0.07	-0.9	9.7	
9	30	60							0	0	0.0	-0.09	-1.7	9.6	
10	30	50							0	0	0.0	-0.12	-1.5	8.5	
11	30	40							0	0	0.0	-0.13	-1.3	7.2	
12	30	30							0	0	0.0	-0.13	-1.1	5.9	
13	30	15							0	0	0.0	-0.20	-0.6	3.3	
14	30	0		999999	999999				0	4500	-1362.8	-0.23	0.0	0.0	
15	0	30							0	0	0.0	-0.02	0.4	6.1	
16	20	30							0	0	0.0	-0.06	-0.2	5.9	
[rad]															
123456															

FIGURE 5: BUILDING EXAMPLE
NODE & MEMBER PLOT, with LOADING



Section 8: Matrix Size Limitations

Microsoft Excel does impose size limitations on matrix operations. The largest matrix that can be inverted with Excel 2000 is 51x51. A 51x51 stiffness matrix limits a problem to 51 Degrees-Of-Freedom (DOF). The FE spreadsheet allows 3 DOF per node (deflection in x & y, rotation in z), thus is limited to 17 nodes by the 51 DOF constraint.

The stiffness matrix is square with size equal to the number of DOF. The number of nodes determines the number of DOF and thus the stiffness matrix size. The number of members is not a factor. Matrices larger than 51x51 may be accommodated with the use of an add-in function such as "Minverse.EXT" (a free add-in available on the internet) allows up to 256x256 matrix, thus allowing an 85-node spreadsheet. Also, more recent versions of Excel may allow larger matrices.

Section 9: Checking Results

It is a good idea to perform some basic checks on results.

- applied forces = calculated reactions
- values are of the expected order of magnitude
- obvious known values (example: a symmetric problem should have symmetric results, a pinned end should have no moment)
- inactive members have no significant force
- deflection of fixed supports approaches zero

It is also a good idea to check critical results.

- The sum of member end forces at a node equals applied node load.
- The sum of member resulting forces equals applied uniform load, and sum of member moments equals zero.

The member force check is particularly useful to verify the sign of reaction forces. It is easy to get mixed up on which direction a member end shear force is acting.

Be wary of thinking that 45 nodes are just not enough for your problems. I have never seen a design that was flawed because the designer used too few nodes. Conversely, I have seen numerous designs that were flawed because the computer model was too large, too complex, and too difficult to visualize and check.

This FE spreadsheet can be an effective tool for analysis, but it can be equally effective in checking output from large complex problems, either by condensing the problem into a small approximation that the spreadsheet can handle, or by analyzing parts from within the large problem.

Section 10: FE Theory Overview

FE method is a numerical solution technique used to analyze continuous systems, in which the system is discretized into a finite number of elements. Continuity of the system is modeled by compatibility equations between adjacent elements. This course will focus on frame-type structures in which the elements are the framing members and the compatibility is of force and deflection. This course gives a basic description of the

solution method used in the FE spreadsheet without going into the details of the underlying theory. More detailed theory can be found from a multitude of sources should the user be so inclined to pursue.

If we limit our scope to members in which

- stress is linearly proportional to strain and
- elements are isotropic, homogeneous members

it follows that member force (**f**) is linearly proportional to member deflection (**d**). Force and deflection for each member can be related by the equation $\mathbf{f} = \mathbf{k} * \mathbf{d}$ where **k** is defined as a stiffness matrix and is determined based on the properties of the member.

$\mathbf{f} = \mathbf{k} * \mathbf{d}$ is to structural engineers what $E=mc^2$ is to physicists. It is the fundamental equation for FE analysis, and once solved can be the key to structural design. In this course, you will learn how to formulate $\mathbf{f} = \mathbf{k} * \mathbf{d}$ for each member, and $\mathbf{F} = \mathbf{K} * \mathbf{D}$ for a system of members. You will learn how to solve for unknowns **f**, **d** and **D**.

I want to emphasize that this course focuses on FE analysis of 2D structural frames subject to static loading, with all elements being linear members with nodes at each end. This is a very specific segment of a huge field of FE applications. FE analysis for this specific segment of problems is really an application of matrix mathematics to solve a series of simultaneous compatibility equations. Some would argue that this is not a true FE analysis since *the system itself is discretized* (a finite number of members connected at a finite number of joints, with shape functions derived by closed form integral calculus).

In the broader realm of FE analysis, the system is generally continuous and *the model is discretized*. Examples would be bending in a flat plate or fluid flow around an obstruction. In these examples, the system is continuous, the model is discretized, and the precision of the solution varies with the refinement of the model. A very fine mesh with a number of small elements will more accurately capture the system behavior than a course mesh.

In a FE frame analysis, dividing members into increasingly small elements by adding intermediate nodes does not increase solution accuracy, rather it has no effect on the solution. This is because member behavior between joint nodes is already solved in close-form solution (for the assumption that only bending and axial strains are relevant). Integral calculus is beyond the scope of this course, but for background information the closed form integral equations for flexure derive from $d^2y/dx^2 = M/EI$.

Since a building frame is essentially a discretized system, a complete solution can be calculated with enough nodes and members. In reality, completely detailing each node and member is impractical and the number is typically pared down to simplify the problem. Examples of some typical frame model simplifications are:

- Modeling a 3D structure in two dimensions.
- Idealizing boundary conditions (“rigid” or “pinned” approximates reality).
- Parsing a model because of symmetry
- Parsing a model for component analysis

- Eliminating / ignoring minor members (ignoring wallboard that is nailed to studs). These assumptions are often appropriate, but a sensitivity analysis may be warranted for verification.

Section 11: FE Spreadsheet Method

Finite element method involves generating a series of compatibility equations relating structure deflections and forces. The equations are handled in matrix form for simplicity and easy of manipulation.

The FE spreadsheet performs the following steps using Excel functions (without macros or VBA programming):

- Calculates member local stiffnesses
- transforms local stiffness to global stiffness components,
- sums global stiffness components to formulate the global stiffness matrix
- inverts the global stiffness matrix
- multiplies the inverted stiffness matrix * node forces to calculate node deflections
- calculates member end forces from node deflections

The basic equation relating local member end-forces with end-deflections is:

(6.1) $\mathbf{f} = \mathbf{k} * \mathbf{d}$, in expanded form becomes:

$$\begin{bmatrix} m_z_i \\ v_y_i \\ v_x_i \\ \hline m_z_j \\ v_y_j \\ v_x_j \end{bmatrix} = \begin{bmatrix} 4EI/L & 6EI/L^2 & 0 & 2EI/L & -6EI/L^2 & 0 \\ 6EI/L^2 & 12EI/L^3 & 0 & 6EI/L^2 & -12EI/L^3 & 0 \\ 0 & 0 & AE/L & 0 & 0 & -AE/L \\ \hline 2EI/L & -6EI/L^2 & 0 & 4EI/L & 6EI/L^2 & 0 \\ 6EI/L^2 & -12EI/L^3 & 0 & 6EI/L^2 & 12EI/L^3 & 0 \\ 0 & 0 & -AE/L & 0 & 0 & AE/L \end{bmatrix} \begin{bmatrix} d_z_i \\ d_y_i \\ d_x_i \\ \hline d_z_j \\ d_y_j \\ d_x_j \end{bmatrix}$$

Bold lettering indicates a matrix.

Lower case matrix letters indicate local coordinates & UPPER CASE letters indicate global coordinates.

The force vector \mathbf{f} consists of m_z , v_y , and v_x for ends “i” and “j”.

The stiffness matrix \mathbf{k} is the 6x6 matrix with variables E, I, L and A.

The deflection vector \mathbf{d} consists of d_z , d_y , and d_x for ends “i” and “j”.

Note that these are “local” matrices, with the x-axis being aligned with the axis of the member.

In global coordinates, the stiffness equations become:

(6.2) $\mathbf{F} = \mathbf{K} * \mathbf{D}$, where:

n = degrees-of-freedom (3*num-of-nodes for the FE Spreadsheet)

\mathbf{F} is a force vector,

\mathbf{D} is the resulting deflection vector,

\mathbf{F} & \mathbf{D} are vectors with “n” number of elements

\mathbf{K} is a square stiffness matrix, of size $n*n$

In a typical structure problem, the known values are:

- member properties (local)
- structure geometry (both local and global), and
- applied forces (global).

Knowing local member properties and geometry we can calculate the local stiffness matrix **k** for each member per Equation 6.1 above.

Knowing the geometric relationship between local and global coordinates for each member allows use to transform each stiffness matrix from local to global coordinates. In the FE Spreadsheet, the local member axis defines the local x-axis, while the global horizontal axis is the X-axis. If the local x-axis is aligned identically with the global X-axis, then local stiffness = global stiffness. For all other cases, where the local member axis is not aligned with the global X-axis, local stiffness does not equal global stiffness. A coordinate transformation is required to calculate the relationship between local and global properties, with that transformation being a function of the angle between the “x” and “X” axes.

In mathematical terms, first define a coordinate transformation matrix, **t** such that:

$$(6.3) \quad \mathbf{d} = \mathbf{t} * \mathbf{D}$$

Member end forces can be similarly transformed by **t** such that:

$$(6.4) \quad \mathbf{f} = \mathbf{t} * \mathbf{F}$$

Recalling (6.1), $\mathbf{f} = \mathbf{k} * \mathbf{d}$,

Then substituting (6.3) and (6.4) into (6.1)

$$\mathbf{t} * \mathbf{F} = \mathbf{k} * \mathbf{t} * \mathbf{D}$$

Then, pre-multiply each side by t-inverse, or \mathbf{t}^{-1} .

$$\mathbf{t}^{-1} * \mathbf{t} * \mathbf{F} = \mathbf{t}^{-1} * \mathbf{k} * \mathbf{t} * \mathbf{D}$$

Simplifying with $\mathbf{t}^{-1} * \mathbf{t} =$ unity matrix and defining $\mathbf{k_T} = \mathbf{t}^{-1} * \mathbf{k} * \mathbf{t}$:

$$(6.6) \quad \mathbf{F} = \mathbf{k_T} * \mathbf{D} \text{ where:}$$

$$(6.7) \quad \mathbf{k_T} = \mathbf{t}^{-1} * \mathbf{k} * \mathbf{t} = \text{a member stiffness matrix transformed to global coordinates.}$$

Once all member stiffness matrices are transformed to a global reference (ie: $\mathbf{k_T}$ is calculated for each member), global stiffness at each node can be calculated by summing the global stiffness of each member that is connected to that node. For example, if members 3, 4 & 5 connect at joint 10, and their X-force stiffnesses are 1, 2 & 3 kips/ft respectively, the global stiffness factor for joint 10 would be the sum of the stiffnesses, or 6 kips/ft. The stiffness of each node is calculated by summing the contributing stiffness elements from each $\mathbf{k_T}$ matrix, then all the node stiffness values are assembled to form the global stiffness matrix, **K**.

Note that if a node is a support point, there is an external or global support stiffness that must be added to the $\mathbf{k_T}$ sum for that node. In the FE spreadsheet, all supports must be entered as support “springs”. As in real structures, there is no infinitely rigid support, but absolute fixity can be approached by inserting very high support spring values.

The known applied global forces are assembled into the force vector, **F**.

Recalling equation (6.2), $\mathbf{F} = \mathbf{K} * \mathbf{D}$; we have now calculated \mathbf{F} & \mathbf{K} so \mathbf{D} is the only unknown. To solve for \mathbf{D} pre-multiply by \mathbf{K}^{-1} , where \mathbf{K}^{-1} = the inverse of \mathbf{K} .

$\mathbf{K}^{-1} * \mathbf{F} = \mathbf{K}^{-1} * \mathbf{K} * \mathbf{D}$, but since $\mathbf{K}^{-1} * \mathbf{K}$ is the identity matrix, this simplifies to
 (6.2) $\mathbf{K}^{-1} * \mathbf{F} = \mathbf{D}$

Inherent in the formulation of the stiffness equations is linear elastic theory, with all its assumptions, particularly that stress and strain are related in a linear fashion, deflections are small and materials are isotropic and homogeneous. Also note that the elements or members in this spreadsheet are limited to axial and flexural strains, shear strains are assumed to be negligible and are not calculated.

FE Method as laid out in the FE Spreadsheet

Figure 6 shows local stiffness and transformation matrices (\mathbf{k} & \mathbf{t}) for each member.

Figure 7 shows:

- Equation (6.7) result, \mathbf{k}_T , for each member.
- Index numbers to position \mathbf{k}_T elements into global stiffness matrix \mathbf{K} .
- Applied load vectors
- Calculated displacements
- Calculated local member forces by substituting (6.3) into (6.5) $\mathbf{f} = \mathbf{k} * \mathbf{t} * \mathbf{D}$

Figure 8 shows:

- \mathbf{K} , global stiffness matrix
- \mathbf{K}^{-1} , inverse of \mathbf{K}
- \mathbf{F} , global applied load vector
- \mathbf{D} , calculated node displacement vector

T_Inverse					
Mem	1	Node	1		
Len	10.00	1	1	1.00	0.00
sin	0.00	1	1	0.00	1.00
cos	1.00	2	2	0.00	0.00
k_xx	700	2	2	0.00	0.00
k_ei	0.2	2	2	0.00	0.00
Uni_Ld	0	2	2	0.00	0.00
Mem	2	Node	2		
Len	7.07107	2	2	1.00	0.00
sin	0.71	2	2	0.00	-0.71
cos	-0.71	3	3	0.00	0.71
k_xx	989.949	3	3	0.00	0.00
k_ei	0.28284	3	3	0.00	0.00
Uni_Ld	0	3	3	0.00	0.00
Mem	3	Node	3		
Len	7.07107	3	3	1.00	0.00
sin	-0.71	3	3	0.00	-0.71
cos	-0.71	1	1	0.00	0.71
k_xx	989.949	1	1	0.00	0.00
k_ei	0.28284	1	1	0.00	0.00
Uni_Ld	0	1	1	0.00	0.00
Mem	4	Node	4		
Len	10	4	4	1.00	0.00
sin	0.00	4	4	0.00	1.00
cos	1.00	2	2	0.00	0.00
k_xx	0.1	2	2	0.00	0.00
k_ei	0.2	2	2	0.00	0.00
Uni_Ld	0	2	2	0.00	0.00
Mem	5	Node	5		
Len	10	5	5	1.00	0.00
sin	0.00	5	5	0.00	1.00
cos	1.00	2	2	0.00	0.00
k_xx	0.1	2	2	0.00	0.00
k_ei	0.2	2	2	0.00	0.00
Uni_Ld	0	2	2	0.00	0.00

$[t]^{-1}$

k_local, Local Stiffness Matrix

	d ir	d iy	d ix	d jr	d jy	d jx
m_l	0	0	0	0	0	0
v_iy	0	0	0	0	0	0
v_ix	0	0	700	0	0	-700
m_j	0	0	0	0	0	0
v_jy	0	0	0	0	0	0
v_jx	0	0	-700	0	0	700
m_l	1	0	0	0	0	0
v_iy	0	0	0	0	0	0
v_ix	0	0	990	0	0	-990
m_j	0	0	0	1	0	0
v_jy	0	0	0	0	0	0
v_jx	0	0	-990	0	0	990
m_l	1	0	0	0	0	0
v_iy	0	0	0	0	0	0
v_ix	0	0	990	0	0	-990
m_j	0	0	0	1	0	0
v_jy	0	0	0	0	0	0
v_jx	0	0	-990	0	0	990
m_l	0	0	0	0	0	0
v_iy	0	0	0	0	0	0
v_ix	0	0	0	0	0	0
m_j	0	0	0	0	0	0
v_jy	0	0	0	0	0	0
v_jx	0	0	0	0	0	0
m_l	0	0	0	0	0	0
v_iy	0	0	0	0	0	0
v_ix	0	0	0	0	0	0
m_j	0	0	0	0	0	0
v_jy	0	0	0	0	0	0
v_jx	0	0	0	0	0	0

$[K_{local}]$

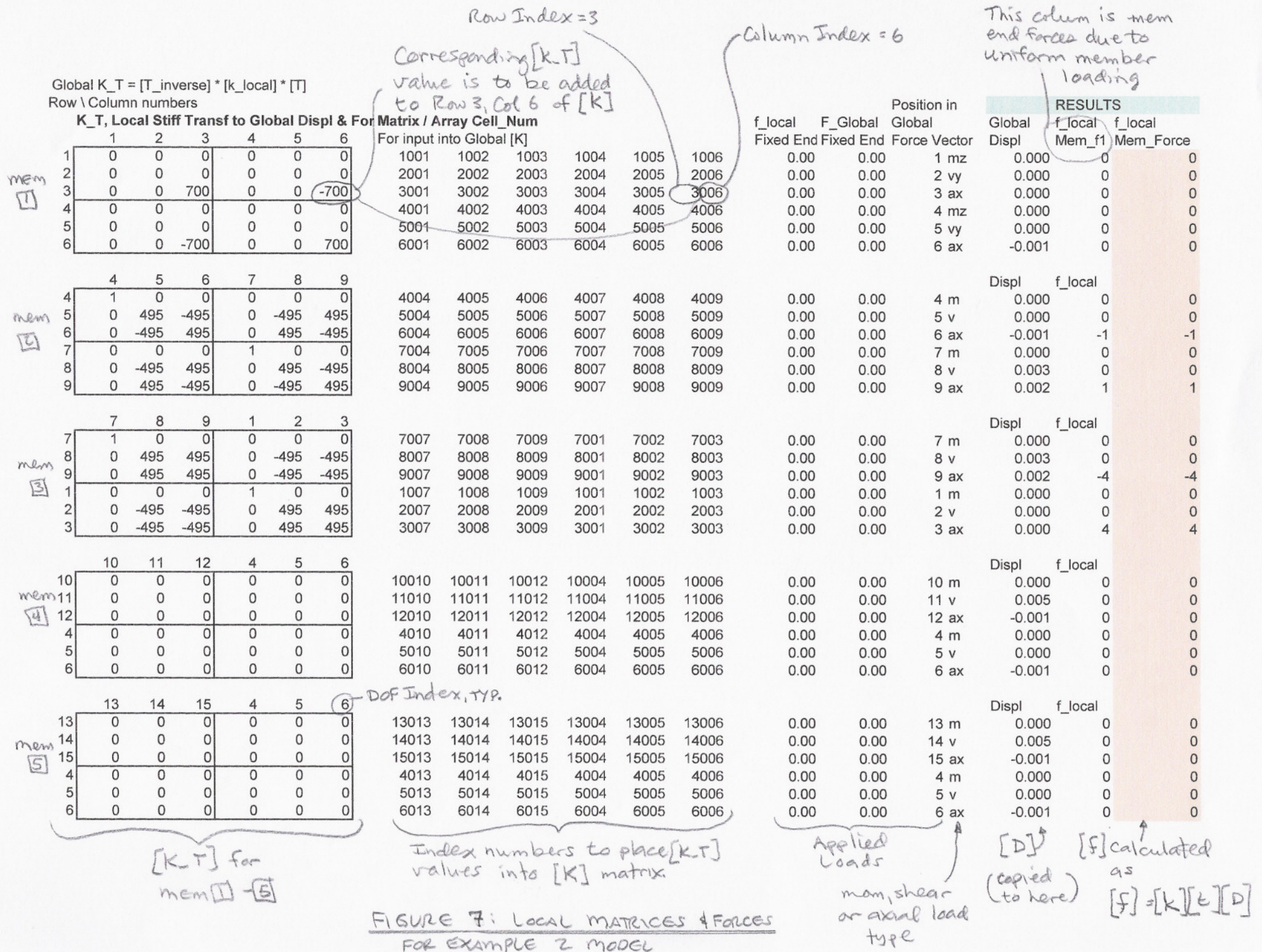
T, Local-to-Global Coord Transf Matrix

1.00	0.00	0.00	0.00	0.00	0.00
0.00	1.00	0.00	0.00	0.00	0.00
0.00	0.00	1.00	0.00	0.00	0.00
0.00	0.00	0.00	1.00	0.00	0.00
0.00	0.00	0.00	0.00	1.00	0.00
0.00	0.00	0.00	0.00	0.00	1.00
1.00	0.00	0.00	0.00	0.00	0.00
0.00	-0.71	-0.71	0.00	0.00	0.00
0.00	0.71	-0.71	0.00	0.00	0.00
0.00	0.00	0.00	1.00	0.00	0.00
0.00	0.00	0.00	0.00	-0.71	-0.71
0.00	0.00	0.00	0.00	0.71	-0.71
1.00	0.00	0.00	0.00	0.00	0.00
0.00	-0.71	0.71	0.00	0.00	0.00
0.00	-0.71	-0.71	0.00	0.00	0.00
0.00	0.00	0.00	1.00	0.00	0.00
0.00	0.00	0.00	0.00	-0.71	0.71
0.00	0.00	0.00	0.00	0.71	-0.71
1.00	0.00	0.00	0.00	0.00	0.00
0.00	1.00	0.00	0.00	0.00	0.00
0.00	0.00	1.00	0.00	0.00	0.00
0.00	0.00	0.00	1.00	0.00	0.00
0.00	0.00	0.00	0.00	1.00	0.00
0.00	0.00	0.00	0.00	0.00	1.00

$[t]$

FIGURE 6: LOCAL STIFFNESS + COORDINATE TRANSFORMATION MATRICES
FOR EXAMPLE 2 MODEL

FROM EQN 6.7: $[t]^{-1}[K][t] = [K_T]$
 \uparrow SEE FIG. 6



Assemble Global [K] by matching Matrix / Array Cell_Num with associated [K] cell and summing in k_T values.

	1	2	3	4	5	6	7	8	9	10	11	12	13	14	15
1	1	0	0	0	0	0	0	0	0	0	0	0	0	0	0
2	0	1000494	495	0	0	0	0	-495	-495	0	0	0	0	0	0
3	0	495	1001194	0	0	-700	0	-495	-495	0	0	0	0	0	0
4	0	0	0	2	0	0	0	0	0	0	0	0	0	0	0
5	0	0	0	0	1000494	-495	0	-495	495	0	0	0	0	0	0
6	0	0	-700	0	-495	1195	0	495	-495	0	0	0	0	0	0
7	0	0	0	0	0	0	1	0	0	0	0	0	0	0	0
8	0	-495	-495	0	-495	495	0	990	0	0	0	0	0	0	0
9	0	-495	-495	0	495	-495	0	0	990	0	0	0	0	0	0
10	0	0	0	0	0	0	0	0	0	0	0	0	0	0	0
11	0	0	0	0	0	0	0	0	0	0	0	0	0	0	0
12	0	0	0	0	0	0	0	0	0	0	0	0	0	0	0
13	0	0	0	0	0	0	0	0	0	0	0	0	0	0	0
14	0	0	0	0	0	0	0	0	0	0	0	0	0	0	0
15	0	0	0	0	0	0	0	0	0	0	0	0	0	0	0
	rot	Dy	Dx												

$[K]$ global

FEM = Fixed End Moment calculated from input U;
F_node = Input force applied at nodes
Defl_Vector = calculated result

	[K-inverse]														
1	1.1412	-1E-07	-1.37E-23	-0.164883	1E-07	-0.0002	-0.2441	0.00019	-0.000121	-0.1648825	1.6488253	-0.000158	-0.164883	1.6488	-0.0001583
2	-1E-07	1E-06	0	-1E-07	-5.4E-26	-1E-22	-1E-07	5E-07	5E-07	-1E-07	1E-06	0	-1E-07	1E-06	1.1097E-22
3	3E-23	-1.62E-25	1E-06	2.43E-23	0	1E-06	0	2.8E-23	1E-06	-2.079E-23	4.436E-22	1E-06	-1.04E-23	3E-22	1E-06
4	-0.165	-1E-07	-1.15E-23	1.141197	1E-07	0.0002	-0.2441	-0.0002	3.72E-05	1.1411968	-11.41197	0.0001583	1.1411968	-11.41	0.00015831
5	1E-07	-1.05E-27	1.08E-25	1E-07	1E-06	1E-22	1E-07	5E-07	-5E-07	1E-07	-2.05E-21	1.1E-22	1E-07	-2E-21	-9.897E-25
6	-2E-04	-9.65E-23	1E-06	0.000158	8.43E-23	0.0014	-2E-20	-0.0007	0.000715	0.0001583	-0.001583	0.0014295	0.0001583	-0.002	0.00142954
7	-0.244	-1E-07	3.17E-23	-0.244072	1E-07	2E-20	1.0059	-1E-20	-0.000131	-0.2440723	2.4407226	-1.33E-20	-0.244072	2.4407	-3.989E-20
8	0.0002	5E-07	-1.79E-23	-0.000191	5E-07	-0.0007	2E-20	0.00137	-0.000357	-0.0001911	0.0019114	-0.000714	-0.000191	0.0019	-0.0007142
9	-1E-04	5E-07	1E-06	3.72E-05	-5E-07	0.0007	-0.0001	-0.0004	0.001369	3.721E-05	-0.000373	0.0007153	3.721E-05	-4E-04	0.00071527
10	-0.165	-1E-07	-2.06E-23	1.141197	1E-07	0.0002	-0.2441	-0.0002	3.72E-05	11.141197	-61.41197	0.0001583	1.1411968	-11.41	0.00015831
11	1.6488	1E-06	1.24E-22	-11.41197	-1E-21	-0.0016	2.4407	0.00191	-0.000373	-61.411967	447.45301	-0.001583	-11.41197	114.12	-0.0015831
12	-2E-04	3.65E-23	1E-06	0.000158	2.16E-23	0.0014	-5E-20	-0.0007	0.000715	0.0001583	-0.001583	10.00143	0.0001583	-0.002	0.00142954
13	-0.165	-1E-07	-2.06E-23	1.141197	1E-07	0.0002	-0.2441	-0.0002	3.72E-05	1.1411968	-11.41197	0.0001583	11.141197	-61.41	0.00015831
14	1.6488	1E-06	1.24E-22	-11.41197	-2.2E-21	-0.0016	2.4407	0.00191	-0.000373	-11.411967	114.11967	-0.001583	-61.41197	447.45	-0.0015831
15	-2E-04	3.65E-23	1E-06	0.000158	2.16E-23	0.0014	-7E-20	-0.0007	0.000715	0.0001583	-0.001583	0.0014295	0.0001583	-0.002	10.0014295

$[K]^{-1}$

Applied Unif. member loads

Applied Node Loads

$[F] =$
Unif. Node Loads

$$[D] = [K]^{-1}[F]$$

FIGURE 8: $[K]$, $[K]^{-1}$, $[F]$ & $[D]$ MATRICES
FOR EXAMPLE 2 MODEL.

Section 12: Linking spreadsheets for progressive analysis

One of the powerful features of spreadsheet FE analysis is the ability to link spreadsheets to represent a series of consecutive loadings. A few examples of this application:

- A steel frame structure that has composite concrete – one spreadsheet could analyze the steel-only loads, a second spreadsheet could analyze loads on the composite section, a third spreadsheet could be a summation of the first two.
- Calculating deflections during a sequential construction of a cantilever constructed bridge
- Analyzing a jacking sequence where the structure is modified in each step, say by adding or removing a member or by jacking at varying points.

A variation on linking is using Excel tables to quantify “what if” scenarios.

Section 13: Benefits, Uses and Limitations of the Finite Element Spreadsheet:

Ease of Use and Minimal Learning Curve:

Most engineers are familiar with, if not adept with spreadsheet software. For those who are regular spreadsheet users, the FE spreadsheet has a minimal learning curve. It only makes sense to incorporate Finite Element analysis into an environment in which so many engineers are already comfortable.

Availability of Spreadsheet and Finite Element Software:

Spreadsheet software is readily available on nearly all engineering computers and can be purchased for a modest (or zero) cost. In contrast, commercial FE software is limited in distribution and high in cost.

With the FE spreadsheet, analysis can be performed on any machine with corresponding spreadsheet software. The spreadsheet is all-inclusive using formulas and Excel functions. There are no macros and no VBA programming, no password protection, no hardware locks, and no license restrictions. In contrast, commercial FE analysis can only be performed on machines with the FE software, and that software nearly always has hardware lock, password or licensing restrictions.

Integration of Data:

Design data for a project is often voluminous and could typically include items such as material properties, structure geometry, loading tabulation, quantity summaries, cost estimates and more. That data is often kept on a series of spreadsheets. As an exception, the actual analysis data from a FE program is typically *not* kept on a spreadsheet. It is common that input-data and output-results from the FE analysis are ported to or from spreadsheets, but the actual analysis and results resides in the FE software.

With the FE spreadsheet, all data, including analysis and results, can all be integrated into one spreadsheet workbook. Consider for example, design of a 3-story building. The designer could have the project design details on a series of spreadsheets (a workbook), such as:

1. Structural framing properties (member properties, material properties, etc)
2. Loads (Structure dead load, live loads, equipment loads, etc)
3. Cost Estimate
4. Finite Element Analysis (member forces, stresses, etc)

The data can all be linked to assure continuity of design in the event of a design change. For example, change the loading of floor-2 from 50 psf to 150 psf on the loading sheet, the FE sheet shows you need to upsize beam weight by 30 plf, the DL weight is increased on that sheet and the cost estimate interactively reflects the new weights.

This is in contrast to using commercial structural analysis software that is not seamlessly integrated into the Excel workbook for the project. Changes need to be updated manually across different programs.

Depth of Data on Spreadsheets:

The FE spreadsheet has all of the input data, output results, intermediate calculations and supporting formulas on one sheet. Each detail can be scrutinized and checked if necessary. This is in contrast to commercial programs that typically do not produce all intermediate steps, nor all underlying formulas.

Ease of Processing Data with Spreadsheets:

Post-processing capability of the FE spreadsheet is superb, with all of Excel's capabilities readily available. In contrast, output from commercial programs is often ported to Excel for the purpose of displaying, formatting, or further using results. With the input, calculations and output all on the same Excel spreadsheet, you can easily format output with colored cells, bold or italic font, you can add arrows or notes, etcetera.

Ease of Modifying Spreadsheet:

An adept and frequent user of the FE spreadsheet may want to customize it for a special need. The FE spreadsheet for structural analysis can easily be linked to other spreadsheets, sequenced with other FE spreadsheets, modified for use in other engineering disciplines, etc.

Ease of Checking:

The FE spreadsheet is all-inclusive with respect to input, equations, intermediate calculation steps and final output. Each step and calculation can be checked in minute detail. In contrast, commercial software is a black-box. You see the input and output, but little of what is in between.

Uses as presented in this course:

- Frame, truss and beam-on-elastic foundation problems.
- Checking large program output, or parts of large program output.

Uses beyond what was presented in this course on enhanced spreadsheets:

- p*y soil-structure interaction
- p*delta analyses
- Excel tables to analyze varying data input
- Seismic response spectrum analysis.

Limitations

- Limited graphics / plotting.
- Limited to 2D frame-type elements (no 3D, no plates).
- Limited to relatively small problems (Approx. 17 nodes with Excel 2000 capabilities, 80 nodes with an add-in matrix inversion function.)
- Expansion of the spreadsheet to vary number of nodes or members requires user manipulation.
- Member end release is not an input variable and must be achieved by manipulation of member local stiffness matrix or inclusion of a short member with small I_{zz} .

Section 14: Summary:

Finite Element method is a powerful analysis tool used by almost all structural design engineers. Many if not most frame-type structural design problems can be solved with the provided Excel spreadsheet. The software is free, easy to use, easy to port between machines, easy to pre- and post-process. In contrast, commercial FE software can be cumbersome to use, difficult to check, difficult to port between machines, and expensive.

More advanced analyses, such as seismic response spectrum problems or soil-structure p-y problems, can also be solved by enhanced versions of the spreadsheet, subject to the size limitations above. Those problems are beyond the scope of this course, but may become available in subsequent courses.

The Excel Spreadsheets provided with this course are believed to be accurate, but use thereof shall be at the users risk. Results of the software should be independently checked (as when making any calculations or using any software). The user is cautioned that the spreadsheet original shall be saved and only copies used for calculations. The formulas in the spreadsheet have minimal protection, so it would be easy to inadvertently (or intentionally) revise the formulas.