FEPC

TRUSS ANALYSIS TUTORIAL

2-D Truss Bridge

February, 2000

PURPOSE OF THE TUTORIAL

This tutorial is designed to guide the beginning student in finite element analysis through a simple truss analysis using the software called FEPC. Step by step instructions are given for defining the model, inputting the data, executing the analysis and evaluating the results. This provides an introduction to the software and its command structure along with use and limitations of truss elements.

After completion of the tutorial you should have a basic understanding of the software operation and use of truss elements in a finite element model. For a more complete description of the software and its capabilities you may read the user's guide by opening the file FEPC33.DOC in a simple text editor such as WORDPAD (It is not a MS WORD file) or open the file FEPC33.WP using WORDPERFECT.

Conventions

To help make the tutorial easier to follow some conventions are defined below. A command or item that needs to be performed is listed on the left. To the right is the description of the action or result of the action.

Fepcip	Type "fepcip" followed by the enter key to start the FEPC input processor. This must be entered on a DOS command line and the program file 'fepcip.exe' must reside in the current directory or be located through the existing PATH.
Fepc 'filename'	Type "fepc" followed by a space and the 'filename'(no ext.) of the model to be analyzed followed by the enter key to start the FEPC solver. This must be entered on a DOS command line and the program file 'fepc.exe' must reside in the current directory or be located through the existing PATH. The file 'filename'.ana must reside in the current directory or may have path designation as part of the 'filename'.
Fepcop	Type "fepcop" followed by the enter key to start the FEPC output processor. This must be entered on a DOS command line and the program file 'fepcop.exe' must reside in the current directory or be located through the existing PATH. You will be prompted for the 'filename' of the model for which results will be displayed.
F2 Model Data	A typical menu command line in the Input and Output Processors. The command is executed by pressing the F2 function key or by using a mouse "left click". These menu selections lead to another menu level or to a prompt to enter data.
Prompt	The software will prompt the user for keyboard input in response to some menu commands. Following the prompt, key in the data requested followed by the enter key. Multiple data entries may be separated by a comma or space.
Mouse	Use the mouse to make command selections from the menu by a "left click". The mouse may also be used for graphic detection in the graphics window using a "left click". When multiple detections are allowed, a "right click" terminates graphic detection input.
↵ó	Press the ENTER key to terminate keyboard input.
⌨	Enter all data requested by the prompt from the keyboard.

Steps in the tutorial will be given in tables such as shown below. The first column gives the function key, the second column gives the command name, and the third column gives a description of the command or more detailed instructions.

F2	MODEL DATA	Select this command to open the submenu for input of model data such as node point locations, element connectivity, loads, boundary conditions, etc.

Analysis of a 2-D Truss Bridge

In the following tutorial, you will design and analyze a given 2-D truss bridge using the FEPC finite element personal computer software. The members of the bridge are assumed to behave as trusses. The tutorial may be outlined in four stages.

I. Input Processing – Create the model using FEPCIP to specify all geometry, boundary supports, loads, and material and section properties, and store an input file for solution.

II. Solving – Execute FEPC to get a static linear elastic solution.

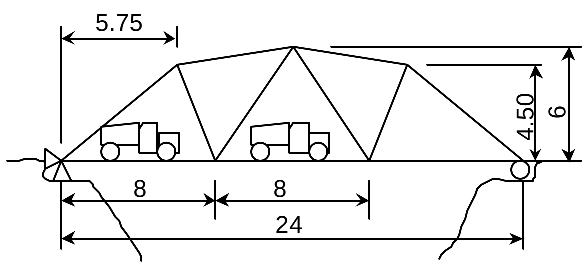
III. Output Processing – View the deformed shape, member loads and stresses using FEPCOP.

IV. Redesign and Reanalysis – Make changes required for a safe design, and reanalyze the model to verify the design changes.

Problem Description

A 2-D Truss Bridge is sketched in the figure that has an overall length of 24 ft. and a maximum height of 6 ft. It is to be made of steel members having a yield strength of 36 kpsi. The lower left corner is assumed to be pinned to the ground while the lower right corner is supported on a roller so that horizontal motion is not constrained. The worst case design load is when one 20,000 lb. vehicle is centered on the left bottom member and another vehicle is centered on the central bottom member.

Suitable cross sections for each member are to be chosen for an overall factor of safety of 4.0 of the bridge.



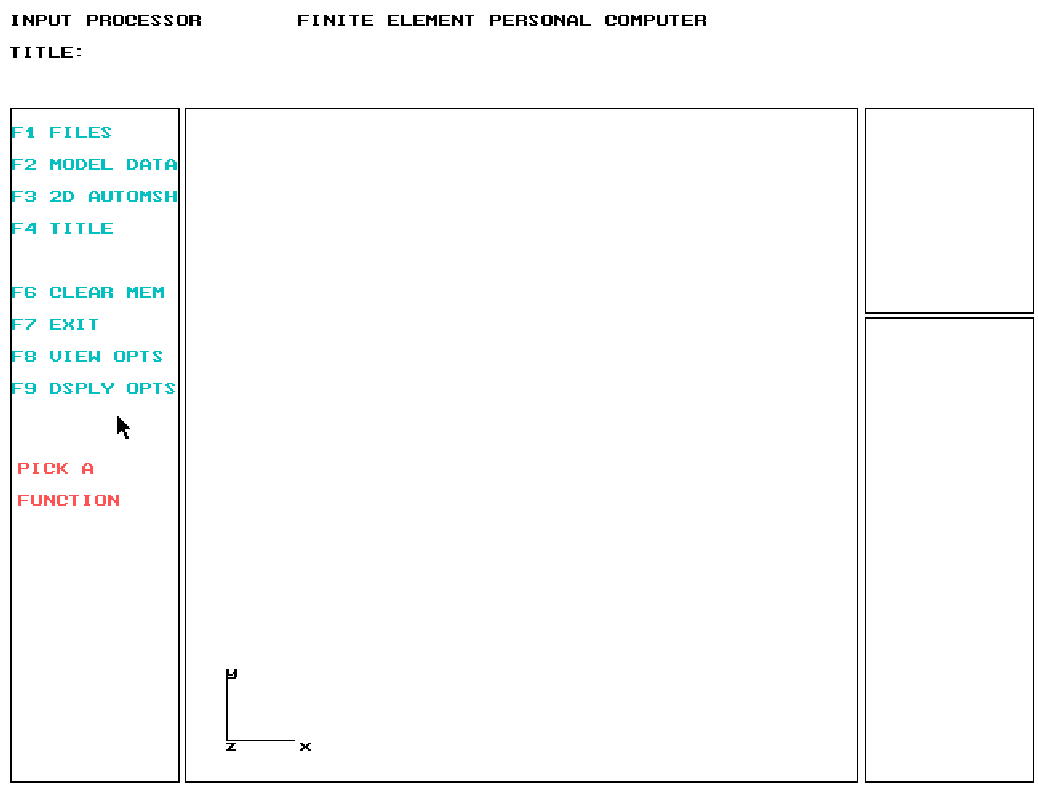
I. Input Processing

Begin the analysis by starting the FEPC Input Processor. From a DOS command line type:

Fepcip

Followed by the enter key. The program file fepcip.exe must reside in the current DOS directory or its directory location must be supplied in the current DOS PATH.

The graphic input screen will open with the display shown on the next page.



F2	MODEL DATA	Left click or press F2 to open the submenu for input of model data.
F1	ELEM TYPE	Click to open the element library menu.
F1	TRUSS	Click to select the truss element for this model.
	prompt	IS TRUSS 2D OR 3D? (Enter 2 OR 3)
	2 ↵ó	
F2	MATL PROP	Click to open the submenu.
F1	INPUT	Click to key in data for the truss element properties.
	prompt	ENTER MATERIAL SET #
	1 ↵ó	There may be more than one material in a model and each element must have a material set #. When elements are created they are automatically assigned the current material set #. This first model will have only one material set.
	prompt	ENTER E, A FOR MATERIAL # 1
	30E6, 1 ↵ó	The modulus of elasticity of steel and a cross sectional area of 1 in^2 are entered. These numerical values for the modulus and area have a units system of inch, pound so the remainder of the model must be consistent. An area of 1 was used so that member loads could be determined and from the loads calculate suitable cross section areas for the members to meet the design goal.
F10	PREV MENU	Click to back up one menu level.

The element has been selected and one set of material and physical properties has been defined. The only appearance change from the startup screen is that current model data is printed in the upper right in the model summary box. It should indicate a 2D TRUSS ELEMENT, 0 NODES, 0 ELEMENTS 1 MATL SET, and the CURRENT MATL IS 1.

This is a good time to introduce file saving. At this point all the model data is in volatile memory of the computer. In order to save the work that has been done, it must be stored to files. Two files will be stored if enough data has been entered. They will have extensions of .mod and .ana using your supplied file name.

If there is not a minimum data set to run an analysis, only the .mod file will be written. The .mod file stores all the input data and is read into FEPCIP when using the recall option. The .ana file is the input file for the solver, FEPC, so it isn't needed until the model is complete.

F10	PREV MENU	Back up to the main menu.
F1	FILES	Left click or press F1 to open the submenu for storing and recalling data files.
F2	STO FN.MOD & FN.ANA	Click to store the current entered data. If you store and exit FEPCIP use the F1 RECALL function to bring the model data back in when you want to continue working on the model.
	prompt	ENTER FILE NAME – NO EXT# (The filename itself may only be 8 characters long due to DOS limitations, but the character string entered may be up to 20 characters long to include any path designations. The 20 characters include the drive letter.)
	Br-truss ↵ó	Enter br-truss or any other file name you choose.
	prompt	BR-TRUSS.MOD file written
	Press any key	
	prompt	NO ELEMENTS DEFINED
	Press any key	
	prompt	Analysis file, br-truss.ANA, not written – incomplete data.
	Press any key	The fn.ana file must be written in order to execute a solution. Be sure it happens on the last file save before exiting FEPCIP.
F10	PREV MENU	Click to back up one menu level.

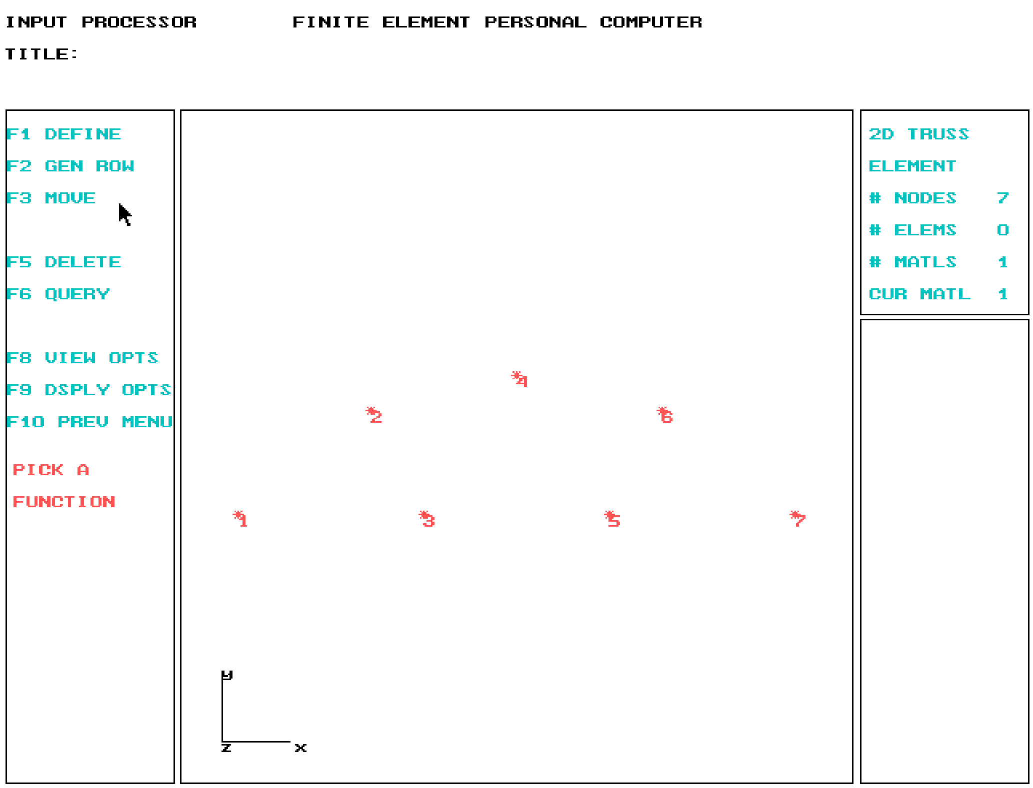
The next step in building the model is to define nodes. Referring to the bridge drawing there are 7 joints in the structure, so there will be 7 nodes in the model. Dimensions in the drawing will specify the node location. As stated before, the numerical values of modulus and area have established an inch, pound unit system. Since the drawing dimensions are given in feet, they must be converted to inches before data entry.

F3	NODE DEF	Left click or press F3 to open the submenu for input of node data.
F1	DEFINE	Click to begin entry of node #'s and coordinates. The node number pattern chosen for this model starts at the left node and proceeds to the right in consecutive order.
	prompt	ENTER NODE #, X, Y, Z =
	1, 0, 0 ↵ó	Choosing the coordinate system origin at the left node, node #1 has x,y = 0,0
	2, 69, 54 ↵ó	If a mistake is made, it may be corrected by using the backspace key before pressing the ENTER key or by defining it again with corrected data.
	3, 96, 0 ↵ó	
	4, 144, 72 ↵ó	
	5, 192, 0 ↵ó	
	6, 219, 54 ↵ó	
	7, 288, 0 ↵ó	
	↵ó	Press the ENTER key with no input to terminate data entry.

At this point all nodes in the model have been defined, but don't appear on screen. This is because the initial window size is 10 x 10 units. Autoscaling will fit all nodes in the window.

F8	VIEW OPTS	Left click or press F8 to open the submenu for scaling the view.
F1	AUTOSCALE	Click to rescale the view so that the entire model fits in the graphics window.
F10	PREV MENU	Click the return to the previous menu.
F9	DSPLY OPTS	By default the node numbers are not displayed. You may wish to turn them on.
F2	LABEL SW	Click or press F2 to open the submenu for display control of node and element numbers or labels.
F1	NODE NOS	
	prompt	OK TO SWITCH ON [Y]#
	↵ó	Press the ENTER key to accept the default Y for yes, or type N for no if the display should not be changed.
F10	PREV MENU	If answer is Y then node numbers will appear on the plot.

Any nodes that appear to be out of position may now be corrected by going back through the F1 DEFINE and entering the correct data. F6 QUERY may be used to check the exact location of any node. Right mouse click terminates query node input. The input screen should now look like the following figure.



All the information needed to define elements is now available. Element definition is done using mouse selection of nodes.

F10	PREV MENU	Left click or press F10 to return to the submenu for input of element data.
F4	ELEM DEF	Click to begin definition of elements. Detection (selection) of two consecutive nodes defines an element. The element material assignment is the current material number and by default is unity when only one material has been defined.
F2	DEFINE	Prompts begin to detect nodes
	prompt	DETECT FIRST NODE
	mouse	Left click near node 1 (mouse will snap to closest node)
	prompt	DETECT SECOND NODE
	mouse	Left click near node 3. Element line is drawn. Element #1 is defined.
	prompt	DETECT FIRST NODE
	mouse	Left click near node 1
	prompt	DETECT SECOND NODE
	mouse	Left click near node 2. Element #2 is drawn.
		Repeat with node pairs (2,3), (2,4), (3,4), (3,5), (4,5), (4,6), (5,6), (5,7), (6,7)
	mouse	Terminate element definition by a right mouse click. Eleven elements should have been created by connecting all nodes in the same geometrical arrangement as in the bridge sketch.

Optional display of element numbers.

F9	DSPLY OPTS	By default the element numbers are not displayed. You may wish to turn them on.
F2	LABEL SW	Click or press F2 to open the submenu for display control of node and element numbers or labels.
F2	ELEMNT NOS	
	prompt	OK TO SWITCH ON [Y]#
	↲ó	Press the ENTER key to accept the default Y for yes, or type N for no if the display should not be changed.
F10	PREV MENU	If answer is Y then node numbers will appear on the plot.
F10	PREV MENU	Return to ELEM DEF menu if any changes are needed.
F10	PREV MENU	Return to MODEL DATA menu to input Restraints and Loads.

The applied displacement boundary conditions are called restraints. These restraints are applied at nodes where the structure support is adequate to prevent any displacement motion. In this case the support restraints are horizontal and vertical at the left node and vertical only at the right node.

F5	RESTRAINTS	Left click or press F5 to open the submenu to define and apply restraints.
F1	SET VALUES	Click to begin prompts to set the restraint values (free or fixed(=0)) to be applied to selected nodes. By default all node restraints are free.
	prompt	SET X-TRANSLATION BOUNDARY CONDITION
F2	FIXED	Left click F2 to set the x-translation displacement value to 0.0
	prompt	SET Y-TRANSLATION BOUNDARY CONDITION
F2	FIXED	Left click F2 to set the y-translation displacement value to 0.0
	prompt	DETECT NODE
	mouse	Left click on node #1. Horizontal and vertical pointing triangles should be

		drawn on node # 1.
	mouse	Right click mouse to terminate node detection and change set values.
F1	SET VALUES	Click to begin prompts to set the restraint values (free or fixed(=0)) to be applied to selected nodes. By default all node restraints are free.
	prompt	SET X-TRANSLATION BOUNDARY CONDITION
F1	FREE	Left click F1 to leave the x-translation displacement value free.
	prompt	SET Y-TRANSLATION BOUNDARY CONDITION
F2	FIXED	Left click F2 to set the y-translation displacement value to 0.0
	prompt	DETECT NODE
	mouse	Left click on node #7. A vertical pointing triangle should be drawn on node # 7.
	mouse	Right click mouse to terminate node detection and end restraint input.
F10	PREV MENU	Return to menu with load input.

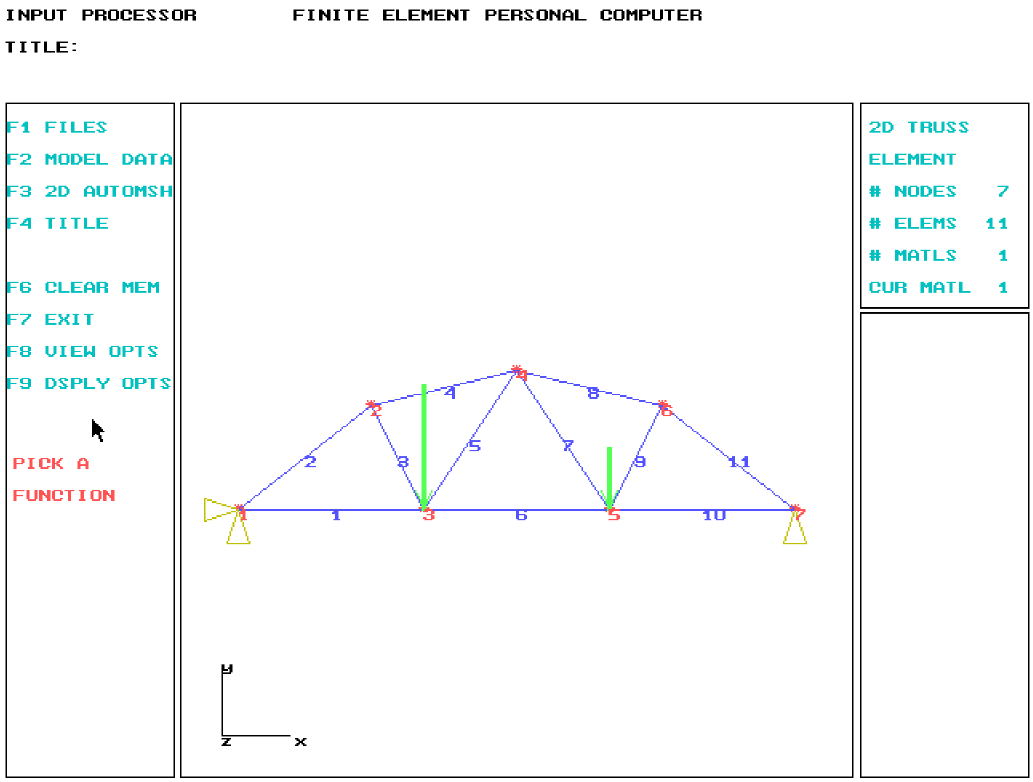
Loads on the bridge are applied by the weight of two 20,000 lb. vehicles. Assuming there are two sides of the bridge, each vehicle applies 10,000 lb. to the side frame. Truss members can not support any load other than axial through the end points. The vehicle loads must be distributed to node points. The vehicle on the left span applies 5000 lb. vertical load on node #1 and 5000 lb. vertical load on node #3. The vehicle on the mid span applies an additional 5000 lb. vertical load on node #3 and 5000 lb. vertical load on node #5. So node #1 carries 5000 lb. vertical load, node #3 carries 10,000 lb., and node #5 carries 5000 lb. vertical load.

If we choose to apply the design factor of safety of 4.0 to the loads then apply 20,000 lb. to node #1, 40,000 lb. to node #3, and 20,000 lb. to node #5. This way the stresses in the members will correspond to the required material yield strength.

Actually, there is no need to apply the load at node #1 since it is restrained vertically and can not move.

F6	LOADS	Left click or press F6 to open the submenu to define and apply loads.
F1	NODE FORCE	Click to begin prompts to set the load values (x and y) to be applied to selected nodes. By default all node loads are zero.
	prompt	ENTER X-FORCE =
	0 ↵ó	Enter 0 for the X-force
	prompt	ENTER Y-FORCE =
	-40000 ↵ó	Enter -40000 for the Y-force
	prompt	DETECT NODE
	mouse	Left click on node #3. The vertical force vector should be drawn.
	mouse	Right click mouse to terminate node detection and change set values.
F1	NODE FORCE	Click to begin prompts to set the load values (x and y) to be applied to selected nodes. By default all node loads are zero..
	prompt	ENTER X-FORCE =
	0 ↵ó	Enter 0 for the X-force
	prompt	ENTER Y-FORCE =
	-20000 ↵ó	Enter -20000 for the Y-force
	prompt	DETECT NODE
	mouse	Left click on node #5. The vertical force vector should be drawn.
	mouse	Right click mouse to terminate node detection.
F10	PREV MENU	Return to MODEL DATA menu and make any needed changes.
F10	PREV MENU	Return to Main menu. You may enter a title at this point
F1	FILES	Use the file store option to save the .mod and .ana files before exiting the program.

The input screen in FEPCIP should now look like the following graphic.



This concludes the INPUT PROCESSING stage of the tutorial.

II. SOLVING

Begin the solution by starting the FEPC processor. From a DOS command line type:

Fepc

Followed by the enter key. The program file fepcip.exe must reside in the current DOS directory or its directory location must be supplied in the current DOS PATH. Some header description of the program is displayed followed by the prompt:

ENTER MODEL FILE NAME – BR-TRUSS

You type in the file name (no ext) as above. The program execution will begin and type out some notes of progress or error messages. Execution may stop when errors occur. The last message for successful execution is “calculating stresses”. If errors occur, the best place to begin is by opening the ‘filename.lst‘ file using any common text editor and evaluate the way the input data has been interpreted. Material properties are a common source of error.

If execution was successful, two additional files will be created for use by the FEPC Output Processor which are fn.msh and fn.nvl. These are binary files only readable by FEPCOP.

III. OUTPUT PROCESSING

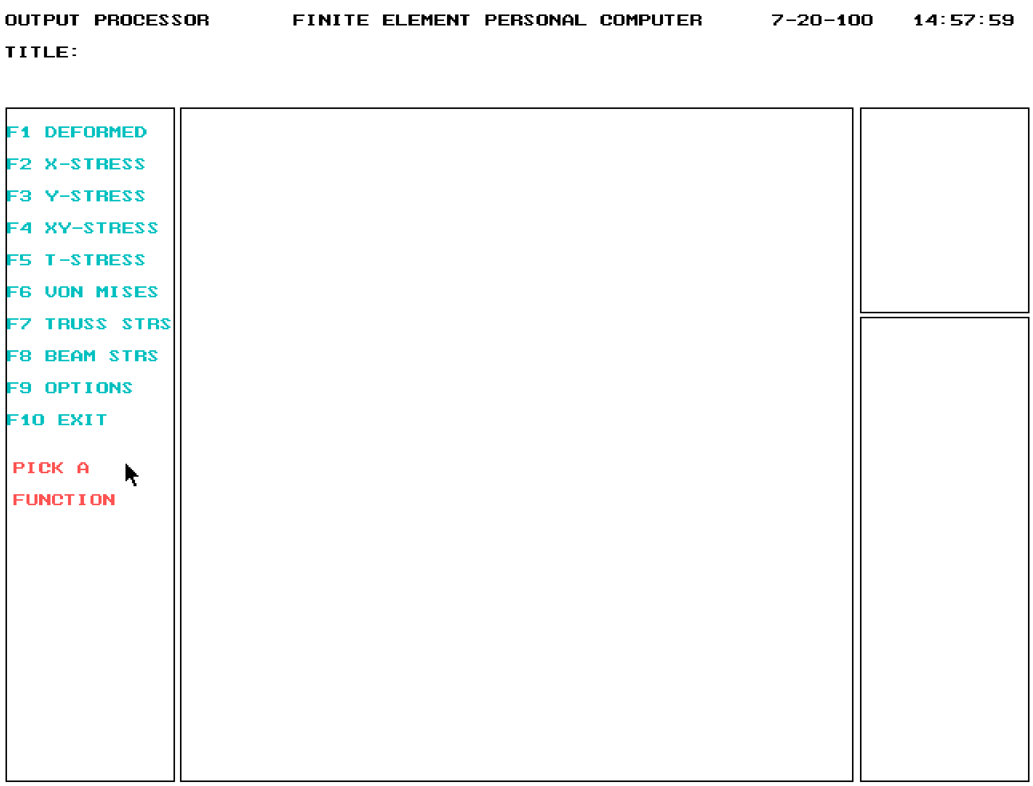
Begin the output processing by starting the FEPC Output Processor. From a DOS command line type:

Fepcop

Followed by the enter key. The program file fepcip.exe must reside in the current DOS directory or its directory location must be supplied in the current DOS PATH. Some header description of the program is displayed followed by the prompt:

ENTER MODEL FILE NAME (NO EXT) – BR-TRUSS

You type in the file name (no ext) as above. The program execution will begin and the graphics window will open in full screen as pictured below.

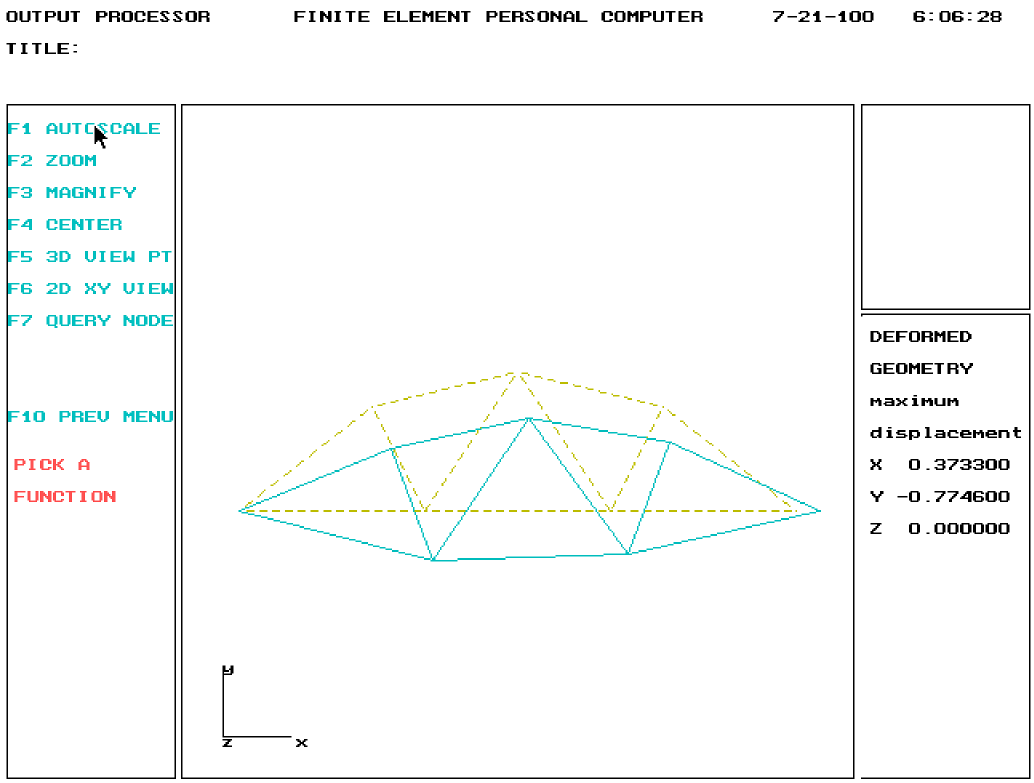


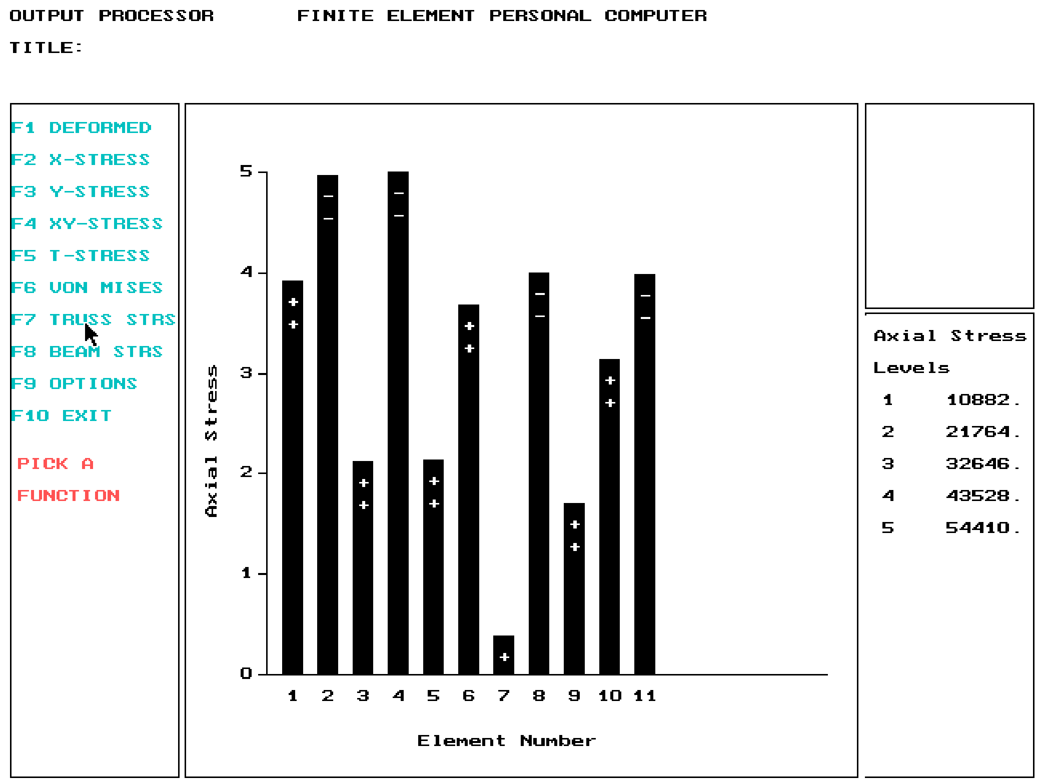
In output processing, the deformed shape and member loads and stresses may be examined. The deformed shape provides opportunity to determine if the shape is realistic based on engineering intuition and to check that boundary conditions and loads have been realistically applied.

F1	DEFORMED	Click or press F1 to open the submenu for display of the deformed shape
F1	PLOT	Click or press F1 to plot the structure in its original and exaggerated deformed shape. The animate function is no longer useful.
F10	PREV MENU	
F7	TRUSS STRS	Click or press F7 to open the submenu for display of member forces or stresses
	prompt	PLOT Force or Stress (F or S)#
⌨	S↵ó	S gives the stress plot, but since all the cross section areas are equal to 1 in^2 the

		forces have the same numerical value. Plus signs on the stress bars indicate tension stressed members and negative signs indicate compression stressed members.
F10	EXIT	Exit Output Processing

The deformed shape and stress plots are pictured below.





IV REDESIGN AND REANALYSIS

The obvious conclusion from an analysis of results is that a 1 in^2 cross section for members is inadequate for the design. Use the yield strength of 36 kpsi to compare with the maximum stress of 54.4 kpsi. As seen in the bar graph of member stresses there are several members with stress exceeding the yield strength. The simple design fix is to increase the cross section area of all members by simple ratio.

$$Area = 54.4 / 36 = 1.51 \ in^2$$

This will bring all members within the allowable stress, but will result in overdesign in some members. Consider a smaller area for members 3, 5, 7, and 9. Since member 5 has the highest stress, size the area based on its stress. Use Notepad or a similar text editor to view the **br-truss.lst** file and find the stress value for member 5 which is 23.27 kpsi. The area is then given by

$$Area = 23.27 / 36 = 0.646 \ in^2$$

Further since common structural steel shapes have only a limited set of sizes, selection must be made using these areas as minimum values. Also, consideration must be given to the shape selection to minimize the possibility of buckling in the compression loaded members. Members 2, 4, 8, and 11 have high compression loads and will all have the higher cross section area.

Use of the Euler Column buckling formula will determine the minimum value of moment of inertia of the cross section to use in selecting the shape. The formula is usually written

$$P_{cr} = \frac{C\pi^2 EI}{\ell^2}$$

where, P_{cr} is the calculated buckling load,
C is the end condition constant,
E is the material modulus of elasticity,
I is the area moment of inertia, and
ℓ is the member length.

Rearranging the formula to yield the moment of inertia gives

$$I = \frac{P_{cr}\ell^2}{C\pi^2 E}$$

Member 2 carries an axial load of 54,090 lb. and has a calculated length of 87.618 in. Using the formula for I yields a value of 1.402 in^4.

Member 4 carries an axial load of 54,410 lb. and has a calculated length of 77.13 in. Using the formula for I yields a value of 1.093 in^4.

Member 8 carries an axial load of 43,530 lb. and has a calculated length of 77.13 in. Using the formula for I yields a value of 0.875 in^4.

Member 11 carries an axial load of 43,270 lb. and has a calculated length of 87.618 in. Using the formula for I yields a value of 1.122 in^4.

Since the same section will be used for each member, the minimum value required is 1.402 in^4.

So the compression members and high stressed tension members will need a section that has a cross section area of 1.51 in^2 with a minimum moment of inertia of 1.402 in^4. It is highly unlikely that such a combination will be found exactly from a finite set of sections, but seek the closest combination for a selection. The remaining tension members (3, 5, 7, and 9) will need a section with a cross section area of 0.646 in^2 without regard to section shape.

Consult the steel section tables in the Appendices of Shigley's Mechanical Engineering Design or other steel section reference to find a suitable selection. A round tubing section of 3 x ¼ has an area of 2.16 in^2 with a moment of inertia of 2.059 in^4. These are approximately 25 % higher than needed. A round tubing of 1 ½ x ¼ has an area of 0.982 in^2 for the smaller members.

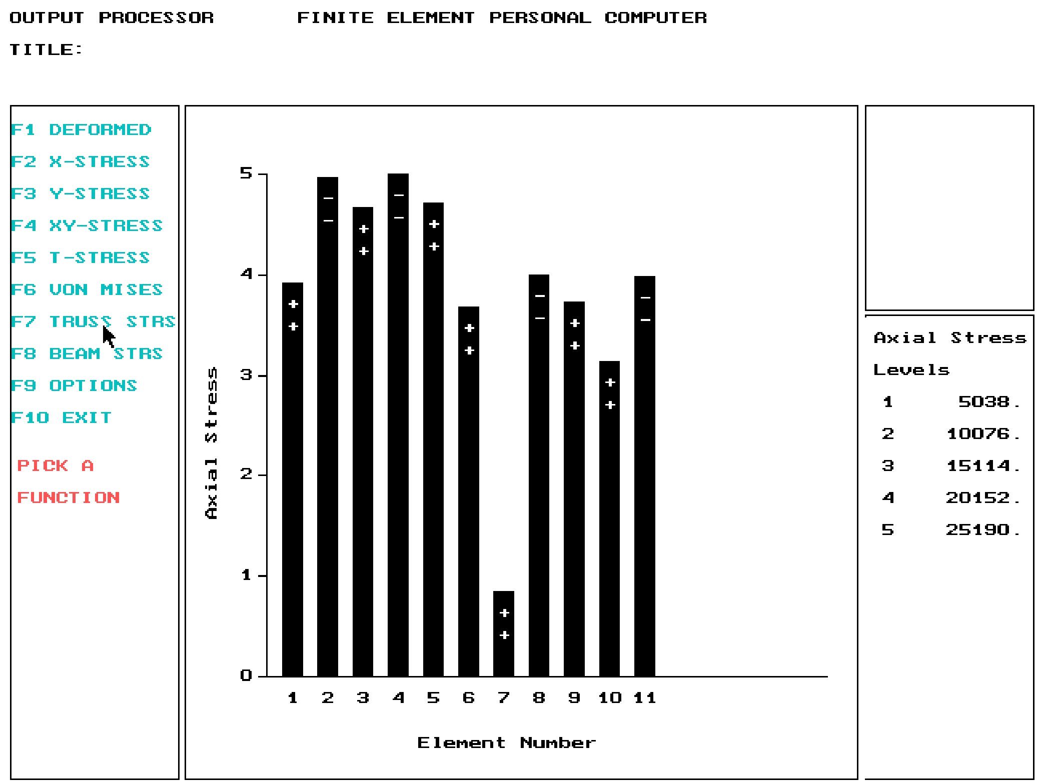
Now to verify these design selections, their properties must be entered into the model and the analysis rerun.

Start up FEPCIP and recall the original model. Then modify the material property set and add a new set and modify the elements that will use the new set.

F1	FILES	Left click or press F2 to open the submenu for file recall.
F1	RCL FN.MOD	Click to bring up the prompt to enter the file name.
	prompt	ENTER FILE NAME – NO EXT#
⌨	Br-truss ↵	The model should appear.
F10	PREV MENU	
F2	MODEL DATA	Left click or press F2 to open the submenu for input of model data.
F2	MATL PROP	Click to open the submenu.
F1	INPUT	Click to key in data for the truss element properties.
	prompt	ENTER MATERIAL SET #

	1 ↵ó	
	prompt	ENTER E, A FOR MATERIAL # 1
	30E6, 2.16 ↵ó	The modulus of elasticity of steel and a cross sectional area of 2.16 in^2 are entered. These values now apply to all the elements, so we must create a second set and modify the appropriate element assignments.
F1	INPUT	Click to key in data for the second material set properties.
	prompt	ENTER MATERIAL SET #
	2 ↵ó	
	prompt	ENTER E, A FOR MATERIAL # 2
	30E6, 0.982 ↵ó	The modulus of elasticity of steel and a cross sectional area of 0.982 in^2 are entered.
F10	PREV MENU	Click to back up one menu level.
F4	ELEM DEF	Click to open the submenu.
F4	MODIFY	Click to select elements to modify.
F1	CHG MATL	
	prompt	ENTER NEW MATL # =
	2 ↵ó	
	prompt	DETECT ELEMENT (Use the mouse to left click on the center of elements 3, 5, 7, and 9. Use a right mouse click to terminate detection.)
F10	PREV MENU	Click to back up one menu level. (You may use QUERY at this level to check the material assignments for any element.)
F10	PREV MENU	Continue to click to top menu level.
F1	FILES	**BE SURE TO SAVE THE MODEL TO THE SAME OR A DIFFERENT FILE NAME BEFORE EXITING FEPCIP!!!**
F2	STO FN.MOD & FN.ANA	Click to store the current entered data
	prompt	ENTER FILE NAME – NO EXT#
	Br-tr2 ↵ó	Enter br-tr2 or any other file name you choose.
F10	PREV MENU	Click to top menu level.
F7	EXIT	Answer prompt

Run the new model using FEPC then run FEPCOP to view the graphical results. Examine the br-tr2.lst file for detailed specific values. Run FEPCOP and enter the filename br-tr2. The deformed shape plot looks the same as before except the maximum displacements have been reduced to 0.172 inches in the horizontal direction and –0.400 inches in the vertical direction. The stress plot is shown below. All stresses are below the material yield strength of 36 kpsi which means that the factor of safety is greater than 4.0 in all members. The actual minimum factor of safety value is the ratio of 36 to 25.19 times 4.0 for a value of 5.72. The redesign has been verified by the finite element analysis.



There is one extremely important aspect of this and every other analysis done by computer assistance that must be considered by the engineer. Never place trust in an analysis without some well done engineering calculations that prove that the analysis is at least close to correct. In this case of a truss analysis, engineering statics can be used to determine some or all of the member loads to compare with the finite element analysis. In all finite element analyses there will always be some kind of engineering approximate solution to make some judgment about the validity of the analysis. Experimental results may also be used effectively to help verify analyses.

The engineer who ignores this advice is eventually doomed to some design catastrophe or at least some highly embarrassing moments.