FEPC

2-D SOLID ANALYSIS TUTORIAL

Notched Rectangular Bar

February, 2000

PURPOSE OF THE TUTORIAL

This tutorial is designed to guide the beginning student in finite element analysis through a simple 2-D solid 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 2-D solid elements.

After completion of the tutorial you should have a basic understanding of the software operation and use of 2-D 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 Notched Rectangular Bar in Tension or Compression

In the following tutorial, you will analyze a given Notched Rectangular Bar for its Stress Concentration Factor using the FEPC finite element personal computer software. The bar is assumed to behave as 2-D elastic. 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.

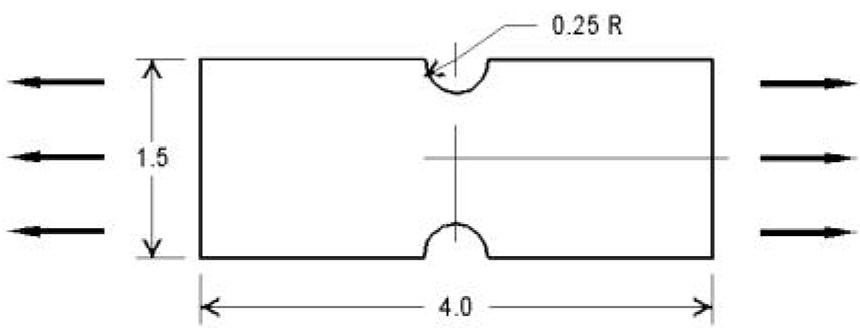
II. Solving – Execute FEPC to get a static linear elastic solution.

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

IV. Re-Mesh and Re-Analysis

Problem Description

The notched rectangular bar in tension or compression is a classic Stress Concentration Factor configuration. This finite element analysis will be undertaken to find a SCF value for a specific geometric configuration and compare with published results. The specific configuration is shown below.



One of the first steps in beginning a typical finite element analysis is to identify the symmetries that may exist and take advantage of them to analyze a reduced size model. In this case there are two planes of symmetry. They are the horizontal and vertical planes through the center of the bar. Symmetry requires that the deformation and stress response be perfectly symmetrical about these planes. So whatever happens in a quarter section of the bar must be mirrored in the other sections. By analyzing one quarter section the full results are known. The requirements to enforce symmetry conditions on the quarter section must be applied in the analysis.

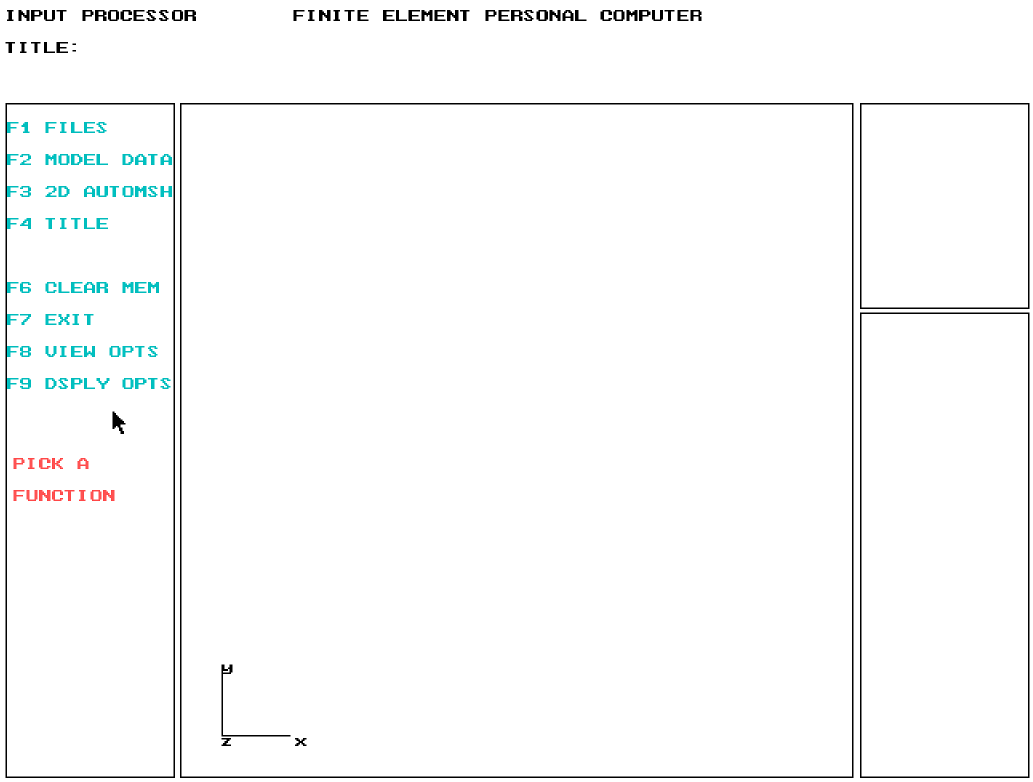
I. Input Processing

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.
F3	PLN STRESS	Click to select the plane stress element for this model. (This assumes the bar thickness is small)
F2	MATL PROP	Click to open the submenu.
F1	INPUT	Click to key in data for the 2-D element properties.
	prompt	ENTER MATERIAL SET #
	1 ↵	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 model will have only one material set.
	prompt	ENTER E, Nu FOR MATERIAL # 1
	30E6, .3 ↵	The modulus of elasticity of steel and Poisson's ratio are entered. This numerical value for the modulus has a units system of inch, pound so the units in the remainder of the model must be consistent.
F10	PREV MENU	Click to back up one menu level.

The element has been selected and one set of material 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 PLN STRESS ELEMENT, 0 NODES, 0 ELEMENTS 1 MATL SET, and the CURRENT MATL IS 1.

This is a good time to review 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.)
⌨	RECTG ↵ó	Enter *rectg* or any other file name you choose. (multiple models must be run to establish convergence of the final solution)
	prompt	RECTG.MOD file written
⌨	Press any key	
	prompt	NO ELEMENTS DEFINED
⌨	Press any key	
	prompt	Analysis file, RECTG.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.

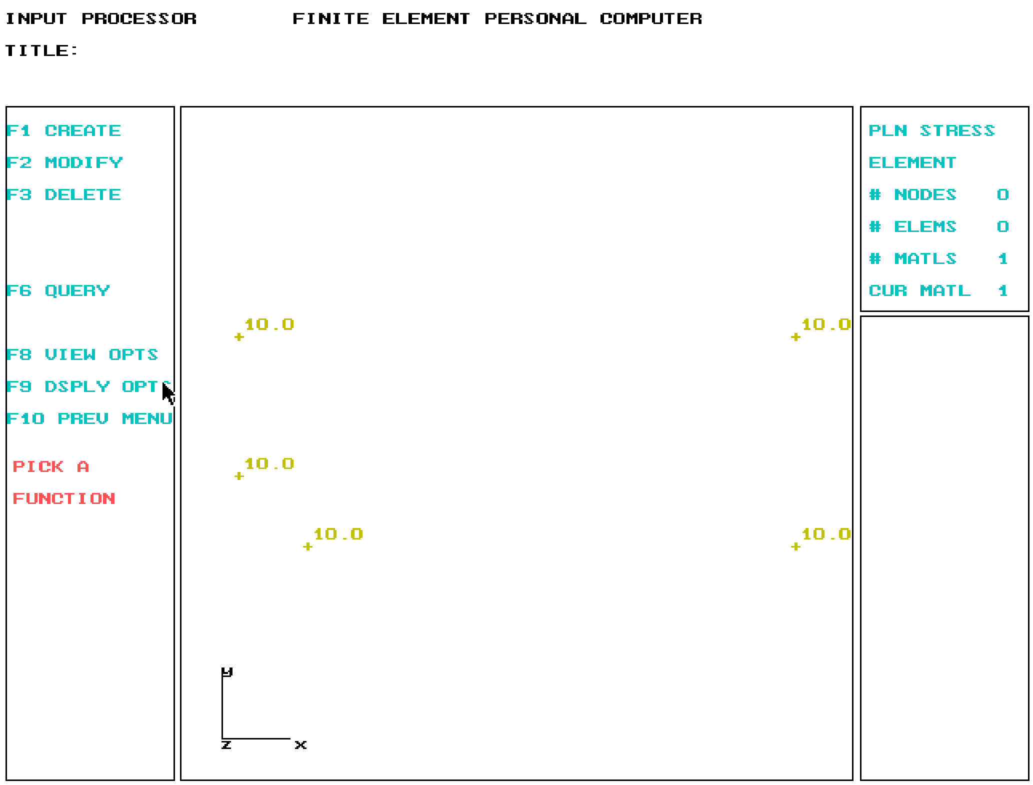
The next step in building the model is to define geometry. The geometry is needed to utilize the node and element mesh generation capability of FEPCIP and avoid the tedious process of entering nodes and elements manually for models with hundreds or thousands of elements.

F3	2D AUTOMSH	Left click or press F3 from the main menu to open the submenu for input of geometry data.
F1	POINT	Click to begin entry of geometry point coordinates.
F1	CREATE	Click for prompt to enter data
	prompt	ENTER X, Y FOR POINT #1 =
⌨	0, 0.75 ↵ó	Starting at the center of the bar enter x,y = 0, 0.75
⌨	0, 0.25 ↵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.
⌨	0.25, 0 ↵ó	
⌨	2.0, 0 ↵ó	
⌨	2.0, 0.75 ↵ó	
⌨	↵ó	Press the ENTER key with no input to terminate data entry.

Now all required points in the geometry to define the straight edges have been defined, but appear on a small part of the screen. This is because the initial window size is 10 x 10 units. Autoscaling will fit all points 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.

Any points that appear to be out of position may now be corrected by going back through the F2 MODIFY and entering the correct data. F6 QUERY may be used to check the exact location of any point. Right mouse click terminates query node input. The input screen should now look like the following figure. The values of 10.0 that appear next to the points are used in the mesh generation stage to control bias spacing of the nodes. More on that later.



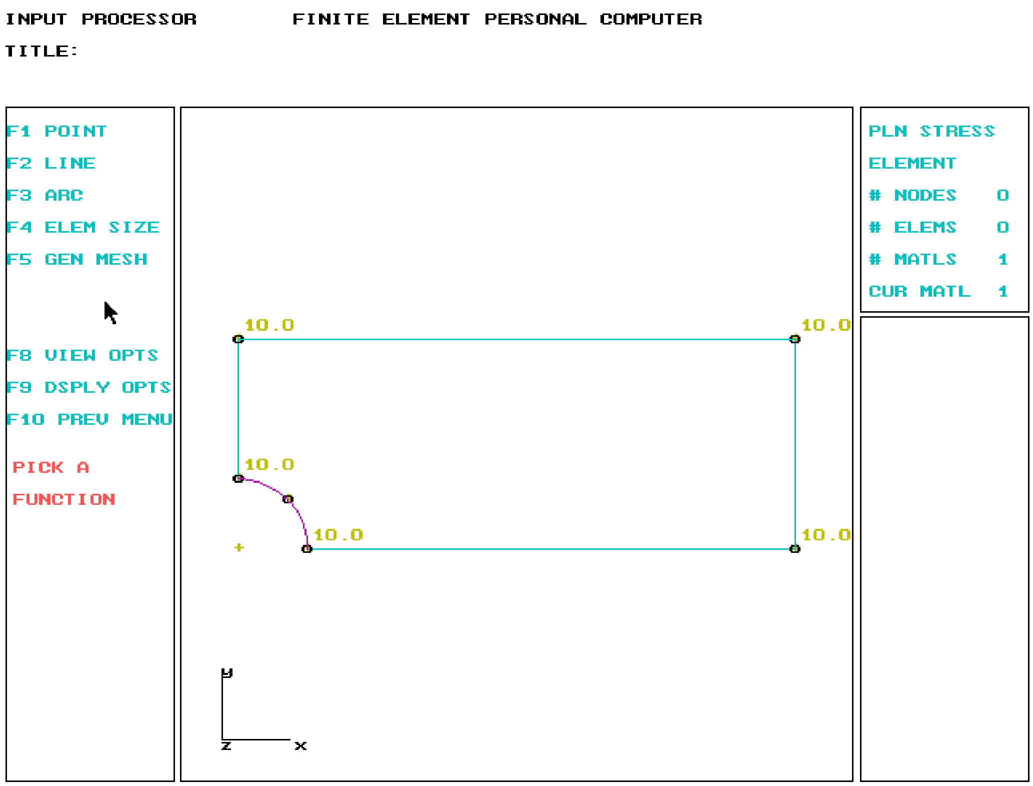
One more point is needed to define the arc. Arcs are defined by three points along the arc, so add a third point. Re-enter the point, create menu and enter a point at 45 degrees along the arc.

F1	CREATE	Click for prompt to enter data
	prompt	ENTER X, Y FOR POINT #6 =
	0.17678, 0.17678 ↵ó	
	↵ó	Press the ENTER key with no input to terminate data entry.
F10	PREV MENU	Left click or press F10 to return to the submenu for input of geometry data.
F2	LINE	Click to begin definition of lines. Detection (selection) of two consecutive points defines a line
F1	CREATE	Prompts begin to detect points.
	prompt	DETECT POINT #1
	mouse	Left click near a point to define a straight line edge (mouse will snap to closest node)
	prompt	DETECT POINT #2
	mouse	Left click near the end point of the line. The geometry edge line is drawn. Continue to draw all four edges of the geometry.
	mouse	Terminate line definition by a right mouse click.

Now define the arc.

F10	PREV MENU	
F3	ARC	Click or press F3 to open the submenu for creating the arc.
F1	CREATE	Prompts begin to detect points.
	prompt	DETECT START POINT 1
	mouse	Left click near a beginning point on the arc in a ccw direction.
	prompt	DETECT MID POINT 2
	mouse	Left click near the mid point of the arc.
	prompt	DETECT END POINT 3
	mouse	Left click near the end point of the arc. The arc is drawn.
F10	PREV MENU	

Now that the geometry is complete, save the file and do not write over it again. This allows building of successive models without re-entering all the geometry again. The screen should look like the figure below.



The next stage is to generate a first mesh for analyzing the model. The principle behind the mesh generation in FEPCIP is to map a grid array of squares into the actual geometry. Please read the section in the user’s guide describing the mesh generation procedure before continuing this tutorial.

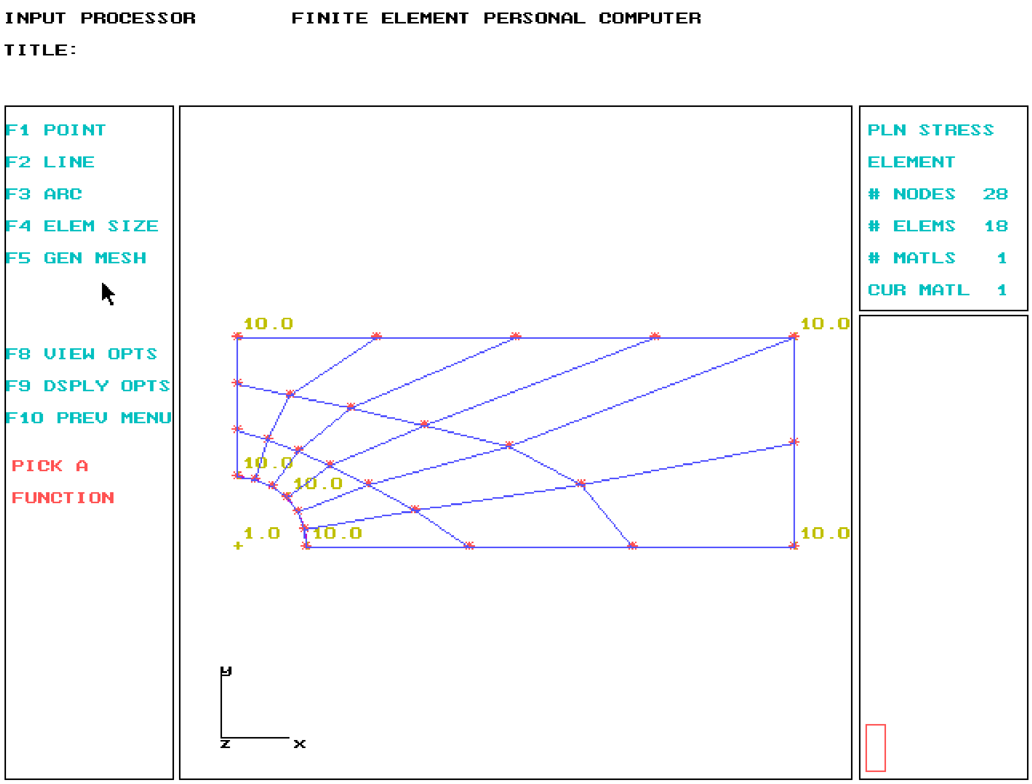
In order to establish a converged solution, a typical finite element analysis begins with a relatively coarse element representation. This provides an initial approximation and helps further guide the modeling process to convergence. It is also much easier to run and debug a small numerical model before proceeding to more converged models.

The first attempt at a mesh will be mapping of a rectangular grid into the geometry. The rectangular grid has four sides while the geometry has five sides, so there must be one shared side. Follow the steps in the next table to build a starting mesh for analysis.

F5	GEN MSH	Click or press F5 to open the submenu for creating the mesh. Prompts begin to detect points
	prompt	DETECT START POINT OF BOUNDARY DEFINITION
	mouse	Left click near the center point of the bar (upper left).
	prompt	Define the mesh boundary by connecting lines and arcs CCW around the region. PRESS A KEY TO CONTINUE When closed press the rgt mouse button or ENTER key at the DETECT LINE prompt. PRESS A KEY TO CONTINUE DETECT LINE OR ARC
	mouse	Left click near the mid point of the line on the left edge of the geometry.
	prompt	ENTER NUMBER OF ELEMENTS #
	3 ↵ó	Try 3 elements along this edge.
	prompt	ENTER DIRECTION #
	1 ↵ó	The integers 3/1 should appear next to the line center.
	prompt	DETECT LINE OR ARC
	mouse	Left click near the mid point of the arc.
	prompt	ENTER NUMBER OF ELEMENTS #
	6 ↵ó	Try 6 elements along this edge.
	prompt	ENTER DIRECTION #
	2 ↵ó	The integers 6/2 should appear next to the arc center. Two red lines should show in the right side lower box on the screen. This represents the outline of the grid of squares being mapped. This box must close for mesh generation to be successful.
	prompt	DETECT LINE OR ARC
	mouse	Left click near the mid point of the bottom edge line
	prompt	ENTER NUMBER OF ELEMENTS #
	3 ↵ó	Try 3 elements along this edge.
	prompt	ENTER DIRECTION #
	3 ↵ó	The integers 3/3 should appear next to the line center.
	prompt	DETECT LINE OR ARC
	mouse	Left click near the mid point of the right side line.
	prompt	ENTER NUMBER OF ELEMENTS #
	2 ↵ó	Try 2 elements along this edge.
	prompt	ENTER DIRECTION #
	4 ↵ó	The integers 2/4 should appear next to the line center.
	prompt	DETECT LINE OR ARC
	mouse	Left click near the mid point of the top side line.
	prompt	ENTER NUMBER OF ELEMENTS #
	4 ↵ó	Try 4 elements along this edge.
	prompt	ENTER DIRECTION #
	4 ↵ó	The integers 4/4 should appear next to the line center, and the red box of the rectangular grid should be closed.
	mouse	Right click to terminate input and begin mesh generation. A mesh should appear with the prompt for MORE MESH GENERATION (Y OR N)# Selecting Y will run more iterations to make the element shapes smoother and selecting N will stop with the displayed mesh. When no further iteration helps select N. Then the prompt.
	prompt	OK TO KEEP (Y OR N)#
	Y ↵ó	This mesh is obviously not a high quality representation of the geometry, but it

		should do a satisfactory job of providing initial results for starting the convergence study.

The screen with the mesh should now look as shown below.



The next step is to apply the displacement restraints to satisfy symmetry and the loads. Symmetry requires that all nodes lying on a plane of symmetry must remain on the plane after loading. Therefore all motion that would take the node off the plane must be restricted.

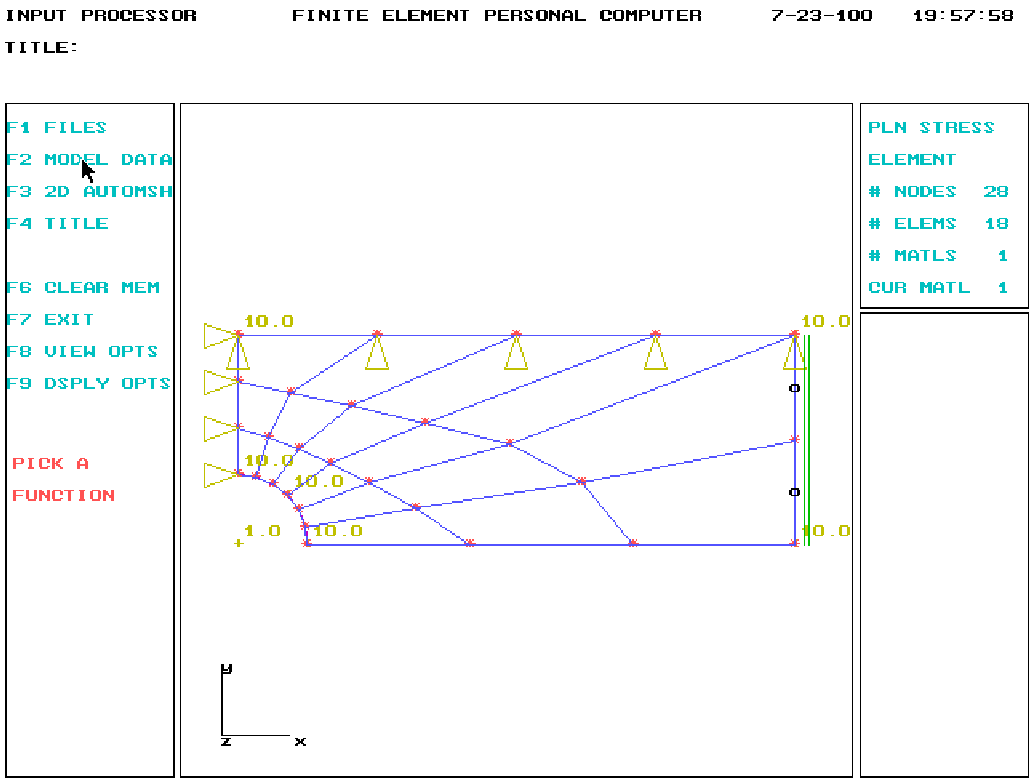
F2	MODEL DATA	Left click or press F2 to open the submenu to apply restraints.
F5	RESTRAINTS	Left click or press F5 to open the submenu to define and apply restraints.
F1	SET VALUES	Click to begin prompts to set the restraint values (free or fixed(=0)) to be applied to selected nodes. By default all node 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 the node at the bar center. Horizontal and vertical pointing triangles should be drawn on the nodes detected.
	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
F2	FIXED	Left click F2 to set the x-translation displacement value to 0.0
	prompt	SET Y-TRANSLATION BOUNDARY CONDITION
F1	FREE	Left click F1 to leave the y-translation displacement value free
	prompt	DETECT NODE
	mouse	Left click on the other three nodes along the left edge symmetry line. Horizontal pointing triangles should be drawn on all nodes detected.
	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 the other four nodes along the top edge symmetry line. Vertical pointing triangles should be drawn on all nodes detected.
	mouse	Right click mouse to terminate node detection and end restraint input.
F10	PREV MENU	Return to menu with load input.

Loading on the model is simple uniform tension or compression along the ends of the bar. The value of load is arbitrary since we are only after the value of stress concentration factor. Therefore choosing a load value to produce unit stress on the net section area will directly yield the stress concentration value as the value of maximum stress in the notch root.

F6	LOADS	Left click or press F6 to open the submenu to define and apply loads.
F2	PRESSURE	Click to prompt to set the pressure value to be applied to selected element edges.
	prompt	ENTER ELEMENT EDGE PRESSURE = (positive values are compression and negative values are tension)
	-0.667 ↵	Enter –0.667 for a tension stress on the right edge that produces an average stress on the net cross section at the center of 1.0 psi.
	prompt	DETECT ELEMENT EDGE
	mouse	Left click on the two element edges on the right side. Parallel lines should be drawn as symbols of the pressure loading.
	mouse	Right click mouse to terminate edge 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. **Name the model RECT1 or some other name different from the geometry file name so that it may be reused for the next model.**

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 – RECT1

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

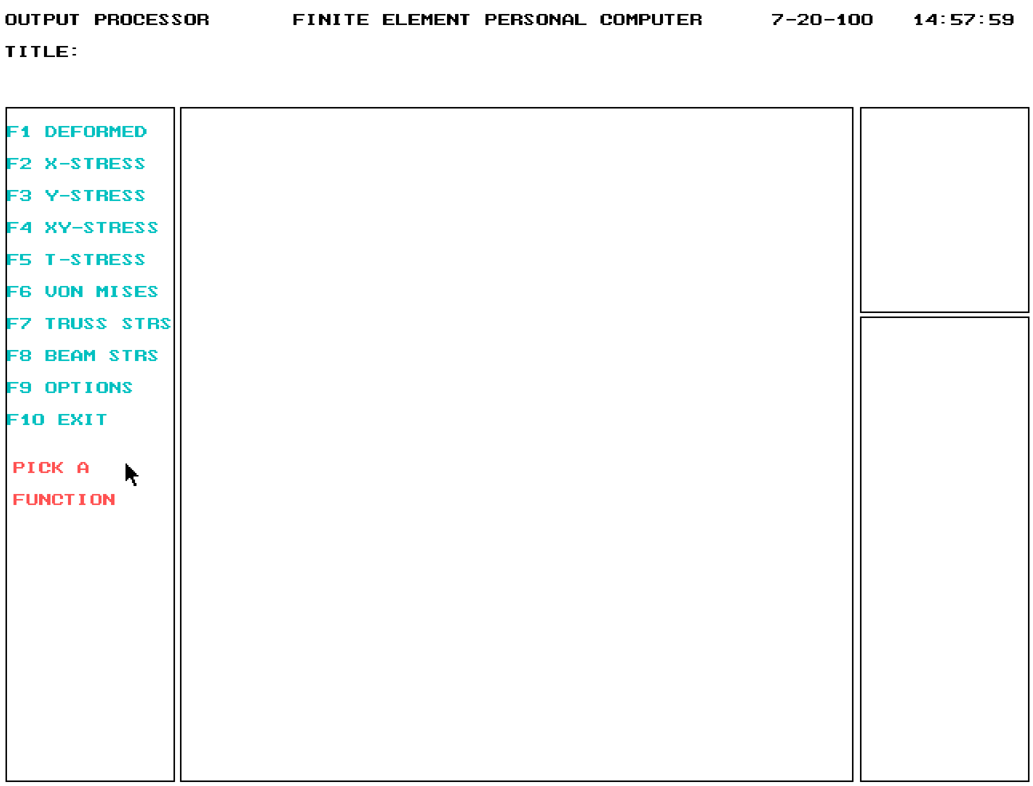
Begin the output processing by starting the FEPC Output Processor. From a DOS command line type:

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) – RECT1

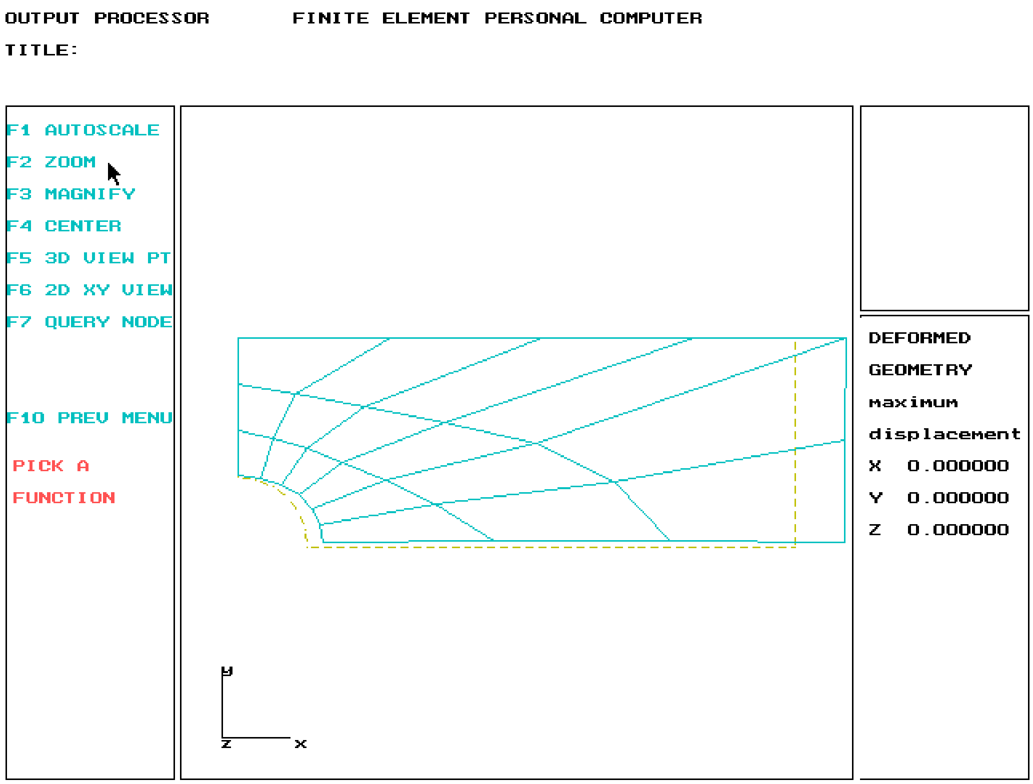
You type in the file name (no ext) as above. The program execution will begin and the graphics window will open in full screen as pictured below.

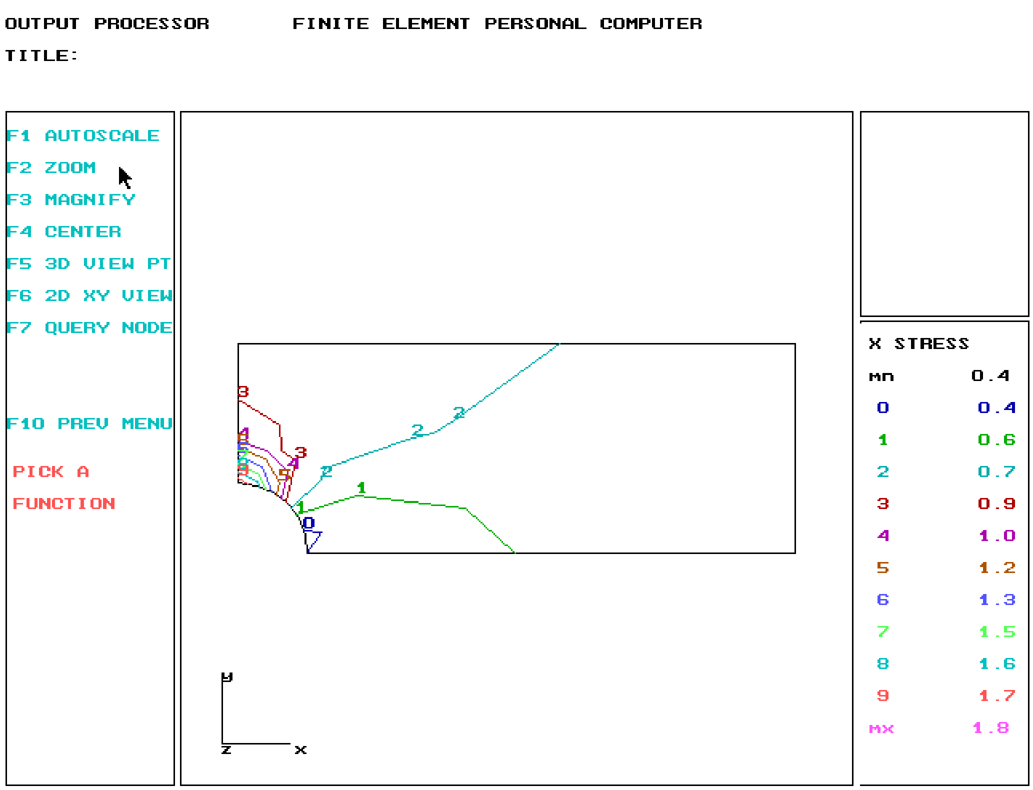


In output processing, the deformed shape and element stresses may be examined. The deformed shape provides opportunity to determine if the shape is realistic based on engineering intuition and to check that symmetry boundary conditions and loads have been realistically applied.

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	
F2	X-STRESS	Click or press F2 to display a contour plot of the x-stress component. This reports a maximum value of 1.8 which is the first value approximation. Since this is a numerical solution there is no known accuracy associated with the first approximation and further solutions must be run to converge the solution. Using the query function gives additional significant figures with a value of 1.842.
F10	EXIT	Exit Output Processing

The deformed shape and x-stress plots are pictured below.





IV RE-MESH AND RE-ANALYSIS

Start up FEPCIP and recall the geometry model. Build a new refined mesh to converge the results toward a more accurate solution. For a good convergence rate, the mesh size should be cur in half as a general guideline. However, more refinement in elements that have high stress gradients and less refinement in elements that have low stress gradients will generally produce faster convergence with fewer overall elements in the refined model.

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#
⌨	RECTG ↵ó	The model should appear.
F10	PREV MENU	
F3	2D AUTOMSH	Left click or press F3 to open the submenu for mesh generation.
F4	ELEM SIZE	Click to set relative element size along a geometry edge. (The size value at the notch root point will be set so that the element size at the root will be half the size at the other end of the geometry left edge and arc.).
	prompt	DETECT POINT #
	mouse	Left click on the point at the notch root.
	prompt	ENTER ELEMENT SIZE (RELATIVE TO 10.0) =
⌨	5 ↵ó	A value of 5 relative to 10 at the top of the left edge makes it half the size at the root, and similarly along the arc.
	mouse	Right click to terminate input.
	mouse	Click view options and autoscale to refresh the display and show the relative size values.

The parameters at now set to generate a new mesh for analysis.

F5	GEN MSH	Click or press F5 to open the submenu for creating the mesh. Prompts begin to detect points
	prompt	DETECT START POINT OF BOUNDARY DEFINITION
	mouse	Left click near the center point of the bar (upper left).
	prompt	Define the mesh boundary by connecting lines and arcs CCW around the region. PRESS A KEY TO CONTINUE When closed press the rgt mouse button or ENTER key at the DETECT LINE prompt. PRESS A KEY TO CONTINUE DETECT LINE OR ARC
	mouse	Left click near the mid point of the line on the left edge of the geometry.
	prompt	ENTER NUMBER OF ELEMENTS #
⌨	6 ↵ó	Try 6 elements along this edge.
	prompt	ENTER DIRECTION #
⌨	1 ↵ó	The integers 6/1 should appear next to the line center.
	prompt	DETECT LINE OR ARC
	mouse	Left click near the mid point of the arc.
	prompt	ENTER NUMBER OF ELEMENTS #
⌨	12 ↵ó	Try 12 elements along this edge.
	prompt	ENTER DIRECTION #
⌨	2 ↵ó	The integers 12/2 should appear next to the arc center. Two red lines should show in the right side lower box on the screen. This represents the outline of the grid of squares being mapped. This box must close for mesh generation to be successful.

	prompt	DETECT LINE OR ARC
	mouse	Left click near the mid point of the bottom edge line
	prompt	ENTER NUMBER OF ELEMENTS #
	6 ↵ó	Try 6 elements along this edge.
	prompt	ENTER DIRECTION #
	3 ↵ó	The integers 6/3 should appear next to the line center.
	prompt	DETECT LINE OR ARC
	mouse	Left click near the mid point of the right side line.
	prompt	ENTER NUMBER OF ELEMENTS #
	4 ↵ó	Try 4 elements along this edge.
	prompt	ENTER DIRECTION #
	4 ↵ó	The integers 4/4 should appear next to the line center.
	prompt	DETECT LINE OR ARC
	mouse	Left click near the mid point of the top side line.
	prompt	ENTER NUMBER OF ELEMENTS #
	8 ↵ó	Try 8 elements along this edge.
	prompt	ENTER DIRECTION #
	4 ↵ó	The integers 84 should appear next to the line center, and the red box of the rectangular grid should be closed.
	mouse	Right click to terminate input and begin mesh generation. A mesh should appear with the prompt for MORE MESH GENERATION (Y OR N)# Selecting Y will run more iterations to make the element shapes smoother and selecting N will stop with the displayed mesh. When no further iteration helps select N. Then the prompt.
	prompt	OK TO KEEP (Y OR N)#
	Y ↵ó	This mesh is a much improved representation of the geometry, and it should do a satisfactory job of starting the convergence study.

F1	FILES	Use the file store option to save the .mod and .ana files before exiting the program. **Name the model RECT2 or some other name different from the geometry file name so that it may be reused for the next model in the convergence sequence.**

The next step is to apply the displacement restraints to satisfy symmetry and the loads. Symmetry requires that all nodes lying on a plane of symmetry must remain on the plane after loading. Therefore all motion that would take the node off the plane must be restricted.

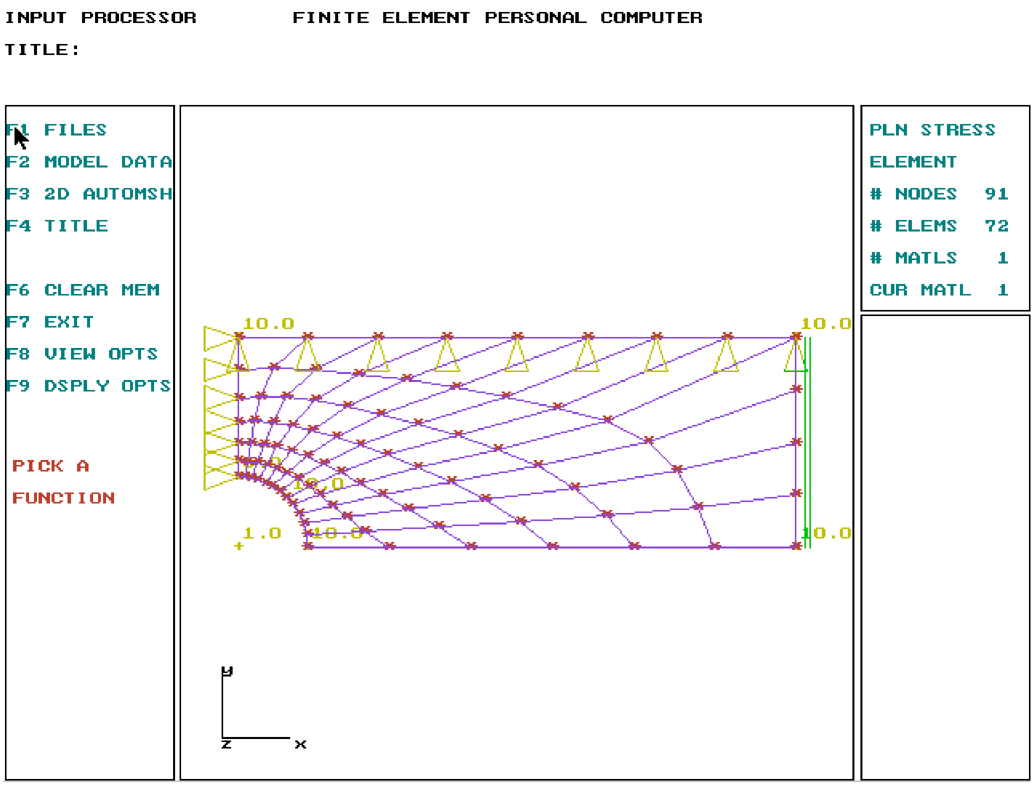
F2	MODEL DATA	Left click or press F2 to open the submenu to apply restraints.
F5	RESTRAINTS	Left click or press F5 to open the submenu to define and apply restraints.
F1	SET VALUES	Click to begin prompts to set the restraint values (free or fixed(=0)) to be applied to selected nodes. By default all node 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 the node at the bar center. Horizontal and vertical pointing triangles should be drawn on the nodes detected.
	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
F2	FIXED	Left click F2 to set the x-translation displacement value to 0.0
	prompt	SET Y-TRANSLATION BOUNDARY CONDITION
F1	FREE	Left click F1 to leave the y-translation displacement value free
	prompt	DETECT NODE
	mouse	Left click on the other nodes along the left edge symmetry line. Horizontal pointing triangles should be drawn on all nodes detected.
	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 the other nodes along the top edge symmetry line. Vertical pointing triangles should be drawn on all nodes detected.
	mouse	Right click mouse to terminate node detection and end restraint input.
F10	PREV MENU	Return to menu with load input.

Loading on the model is simple uniform tension or compression along the ends of the bar. The value of load is arbitrary since we are only after the value of stress concentration factor. Therefore choosing a load value to produce unit stress on the net section area will directly yield the stress concentration value as the value of maximum stress in the notch root.

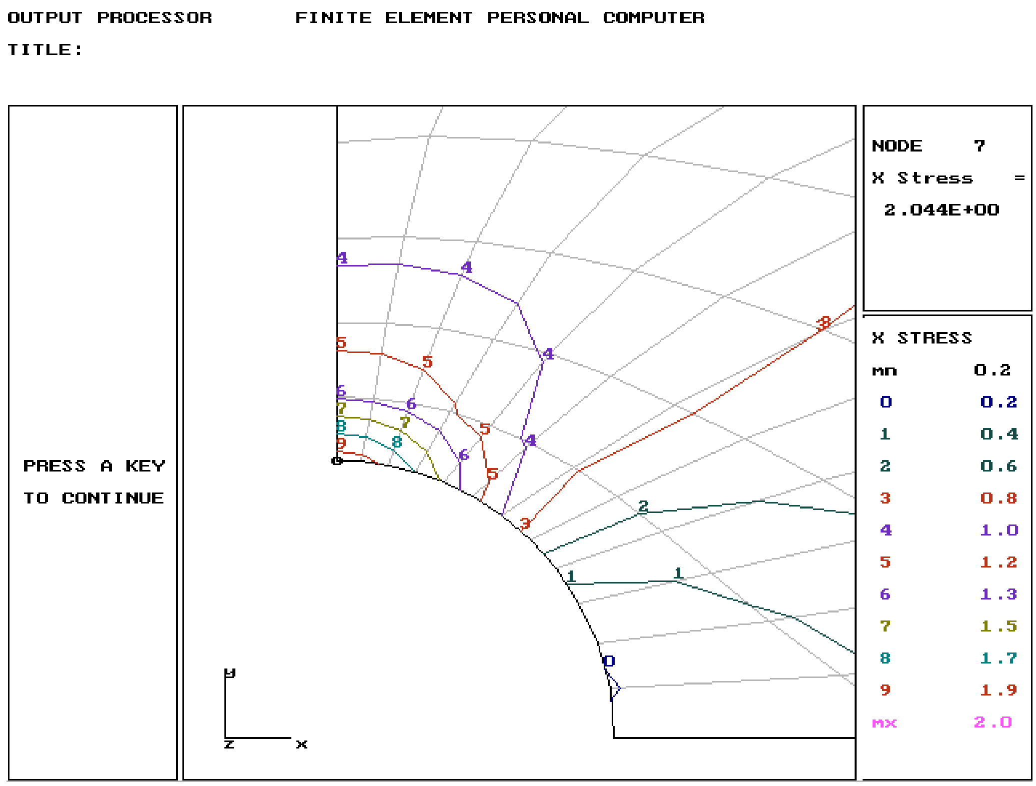
F6	LOADS	Left click or press F6 to open the submenu to define and apply loads.
F2	PRESSURE	Click to prompt to set the pressure value to be applied to selected element edges.
	prompt	ENTER ELEMENT EDGE PRESSURE = (positive values are compression and negative values are tension)
	-0.667 ↵	Enter –0.667 for a tension stress on the right edge that produces an average stress on the net cross section at the center of 1.0 psi.
	prompt	DETECT ELEMENT EDGE
	mouse	Left click on the element edges on the right side. Parallel lines should be drawn as symbols of the pressure loading.
	mouse	Right click mouse to terminate edge 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. **Name the model RECT2 or some other name different from the geometry file name so that it may be reused for the next model.**

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



Run the new model using FEPC then run FEPCOP to view the graphical results. The deformed shape plot verifies that the boundary conditions have been applied properly and that the shape is reasonable. The X-stress plot in zoomed in view with element outlines turned on is shown in the next figure. This illustrates that the new mesh is refined appropriately in areas of high stress gradient. However, in this stress concentration problem there is still a high stress gradient across the single element at the notch root indicating that further refinement is needed. Indeed there is no guarantee or assurance of accuracy in any numerical solution when there are only two results. The value changed from 1.842 to 2.044. The published value is 2.2 in this case so the results are converging. However, generally a finite elment analysis is done to find the unknown result accurately. Therefore proper convergence of solutions is imperative.

This concludes the tutorial, however, the student motivated to learn should continue the convergence process until an accurate solution is reached and read the text to determine how the overall mesh and results should be interpreted and evaluated.



ENGRAVE THE FOLLOWING IN YOUR BRAIN!

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.