

# MANUAL for e-Handbook

## 1 Quick tour of Windows, Menus and Tool Bars

The majority of the time in the fracture analysis is spend on the creation of input files. The e-Handbook of Fracture is a Java based front end for the preparation of the input files (pre-processing). It also allows us to run the Fortran program (main-processing) and read off stress intensity factors from crack tips present (post-processing). The package is available for download at <http://rci.rutgers.edu/~denda/e-Handbook>. The instructions for installation to your computer and other pertinent information can be found at the web site.

Figure 1 shows a snapshot of the initial appearance of the e-Handbook as you open the program. The center blue piece of the e-Handbook is the **Graphic Editor** window, and to the left of it is the vertical **Tool Bar**. At the top of the **Graphic Editor** window, where the **Graphic Editor** tab is placed, two more tabs will be added later to display the input and output files. The **Tool Bar** contains several tools represented by icons, which can be identified by hovering the mouse over them. These tools are grouped into pre-processing (initialize, refine and edit) tools, main-processing (save and run) tools, and post-processing SIF tool. In addition we find a **Menu Bar** on top, which contains standard Windows File functions and other items to be described below.

### 1.1 Graphic Editor

Before the input file is created, the e-Handbook displays only one tab for the **Graphic Editor**. This is the default window shown at the start of the e-Handbook. Horizontal and vertical rulers are shown by default and can be switched on and off through **Options** menu using **Show Ruler** checkbox. The default **Graphic Editor** cursor is a cross hair sign (+), aided by a horizontal and a vertical lines through the cursor for better positioning of coordinates if the **Show Ruler** option is turned on. These two lines remain whether the ruler is present or not.

### 1.2 Tool Bar

Items in the **Tool Bar** are arranged to follow the work flow of pre, main and post processings (Figure 2). The **Material Properties** tool is a single member of the first group. The second group is the Element Creation Group, containing the **Add Boundary**, **Add Center Crack**, **Add Edge Crack**, and **Add Observation Point** tools. The third group has tools for modifying and fine-tuning the created elements, which includes the **Refine Element**, **Merge Boundary**, **Reverse/Delete Boundary** and **Edit Geometry** tools. The fourth group has tools for viewing the geometry, which includes the **Pan**, **Zoom In**, **Zoom Out**, and **Zoom Default** tools. The fifth group has two tools, the **Save** tool to produce the input file and the **Run** tool to

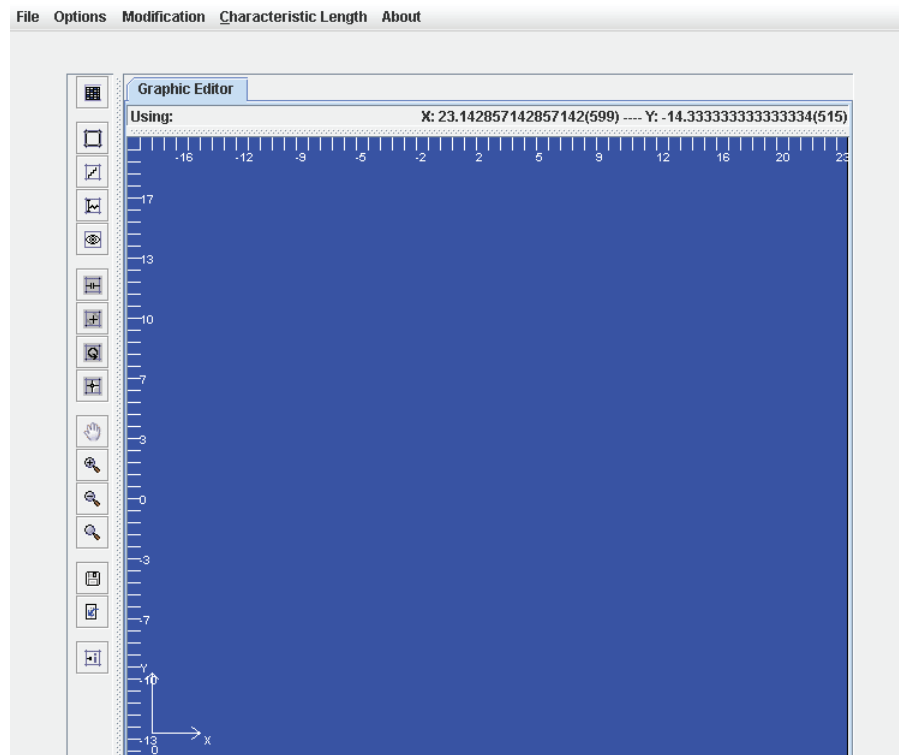


Figure 1: e-Handbook of Fracture startup with Graphic Editor window (center), Tool Bar (left) and Menu bar (top).

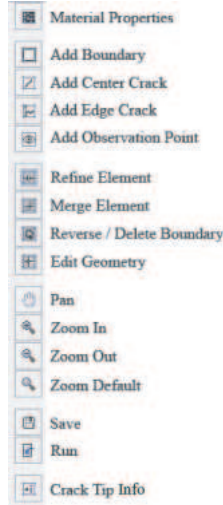


Figure 2: Tool Bar

run the Fortran program. The last group has the **Crack Tip Info** tool, which is only accessible after the **Run** tool is executed.

Although these tools are arranged in the order input files are prepared, there is no requirement to strictly follow the order from the top to bottom. Once the coarse boundary is created either by creating a simple boundary or through merging of multiple simple boundaries, it is advisable to use the **Refine Element** tool to refine the mesh and apply the boundary conditions. The **Edit Geometry** Tool is used to modify the coordinates of any of the boundary and cracks elements. For example, the kinked crack is produced by moving the mid nodes out of the original straight alignment configuration with this tool. On the other hand, the **Run** tool can be executed only after the input file is created, either by saving the current mesh or loading an existing input file. Notice that the name of the current tool or instructions on how to use it is shown in the **Information Bar** located under the tabs and above the blue **Graphic Editor** screen.

### 1.3 Menu Bar

The **Menu Bar** contains five menu items (Figure 3). The **File** menu contains the **New**, **Save**, **Load** and **Exit** commands. Use the **New** command to delete the contents of the current **Graphic Editor** window to start a new session. The **Save** command is used to save the contents of the current **Graphic Editor** into an input file for the main-processing or to store the current mesh configuration to be completed later. If you have an existing input file, then use the **Load** command to load the file into the e-Handbook; a contour with the specified geometry and boundary condition will be drawn on the **Graphic Window** and the material properties are loaded. The **Load** command is handy for checking the integrity of input file created manually elsewhere.

The **Options** menu contains three checkboxes: **Show Node Numbers**, **Show Ruler** and **Show All**



Figure 3: Menu Bar

**Boundary Conditions.** These checkboxes are on by default, but they may be turned off in case the **Graphic Editor** will get too clustered. The **Modification** menu provides an alternative way to modify the individual element that can also be performed by the **Edit Geometry** tool. While the latter relies on mouse clicks the former primarily uses text input for specifying the nodal coordinates and double-node properties of the elements. The user can also undo any unwanted modifications through the **Undo Modify Boundary** and **Undo Modify Crack** commands.

The **Characteristic Length** menu is used to select the length scale of the problems. By selecting the proper characteristic length suitable for each problem, we can deal with scales that ranges from nanometer to kilometer. It is also possible to specify the custom characteristic length not listed in the selection. Notice that the non-dimensional input data prepared by the e-Handbook is used by the main-processor and the results are reported in the dimensional quantities using the relations (??). Input of the characteristic length  $x_0$  makes this possible. If the characteristic length is not specified, then the input file is prepared with the default characteristic length of 1 meter.

## 2 Input file preparation

### 2.1 Characteristic length

The characteristic length determines the appropriate length scale of the problem you are working with. Go to the **Characteristic Length** menu and select an appropriate length scale for the problem. For example, if you are dealing with a MEMS problem, then click the Micro radio button. If the problem is for the faults in the ground, then click the geo radio button. Notice that the subsequent entry for the contour and crack geometry is given in non-dimensional coordinates based on the selected characteristic length. Computation is performed with non-dimensional variables and the final output is given in the dimensional form.

### 2.2 Material constants

The material constants are specified using the **Material Properties** tool located at the top of the **Tool Bar**. Click the icon and a material selection window opens (Figure 4). Select the appropriate material from the drop down list. Notice the values of compliances listed in the list are dimensional. The e-Handbook will non-dimensionalize these constants by dividing them with  $s_0 = 10^{-11}(m^2/N)$ .

It is possible to input custom material constants not listed in the look up table. Edit the first two lines in the Comments box: name of the material (first line) and the crystal class (second line), any text inserted

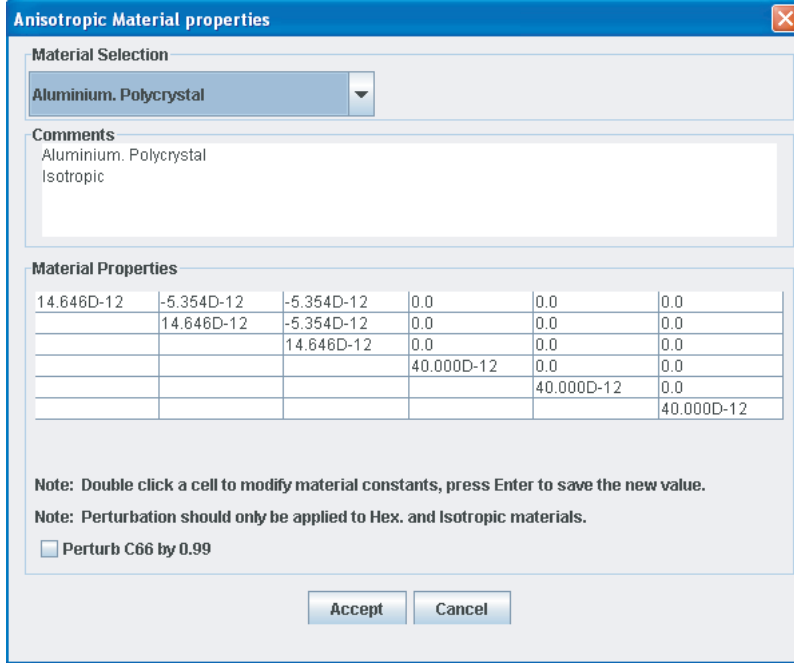


Figure 4: Material Properties window

after the second line will simply be ignored when generating the input file. Edit the entries of the Material Properties matrix (upper triangle only) to enter the custom values by double-clicking the desired cell. Make sure you have typed in the proper dimension of the coefficients. For example, material constants for the isotropic materials with Young's modulus  $E$  and Poisson's ratio  $\nu$  are given by

$$S_{iso} = \begin{pmatrix} s_{11} & s_{12} & s_{13} & s_{14} & s_{15} & s_{16} \\ & s_{22} & s_{23} & s_{24} & s_{25} & s_{26} \\ & & s_{33} & s_{34} & s_{35} & s_{36} \\ & & & s_{44} & s_{45} & s_{46} \\ & & & & s_{55} & s_{56} \\ & & & & & s_{66} \end{pmatrix} = \begin{pmatrix} \frac{1}{E} & \frac{-\nu}{E} & \frac{-\nu}{E} & 0 & 0 & 0 \\ & \frac{1}{E} & \frac{-\nu}{E} & 0 & 0 & 0 \\ & & \frac{1}{E} & 0 & 0 & 0 \\ & & & \frac{2(1+\nu)}{E} & 0 & 0 \\ & & & & \frac{2(1+\nu)}{E} & 0 \\ & & & & & \frac{2(1+\nu)}{E} \end{pmatrix} \quad (1)$$

Since the three characteristic roots  $P_\alpha$  ( $\alpha = 1, 2, 3$ ) of the isotropic materials are equal to  $i = \sqrt{-1}$  (imaginary number), the assumption of distinct roots are violated. All we have to do is to perturb the values of some of the components in (1) to produce three distinct characteristic roots. Usually it is sufficient to change the value of only one component to produce three distinct roots. We recommend perturbing  $s_{66}$  by 1%, which can be accomplished by clicking the Perturb checkbox to have the data modified for you. Although users can select any other custom perturbations, it is not recommended to perturb more than 1% of each value. The similar perturbation is required for other anisotropic materials, such as transversally isotropic (i.e., hexagonal) materials, for which the characteristic roots are also not distinct. If the final results, in the **Output Window**, are in doubt, then check the characteristic values (or eigenvalues) printed in the

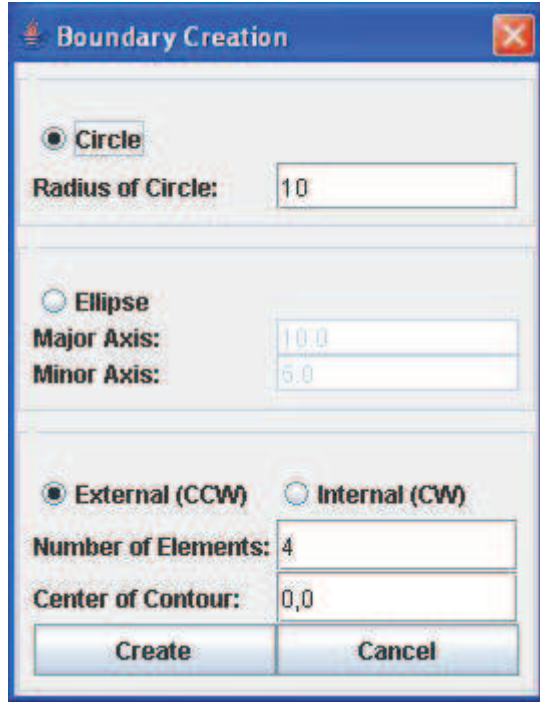


Figure 5: Boundary Creation window

**Output Window.** If two or three characteristic roots are nearly identical, then try to increase the amount of perturbation and retry the calculation with the new material constants.

### 2.3 Boundary geometry

Boundary Creation: For the simply connected region, the unmeshed boundary is introduced by the **Add Boundary** tool in the **Tool Bar** (Figure 5). The e-Handbook generates a polygon enveloped either by a circle or an ellipse. First select either the **Circle** or **Ellipse** radio button. If the **Circle** is selected, then specify the **Radius of Circle**, the default is 10 in the unit specified as the characteristic length. For the **Ellipse**, specify the **Major Axis** and **Minor Axis**. Next specify whether the created boundary will be an **External** (counter-clockwise) or **Internal** (clockwise) boundary. Notice that the external boundary has the counter-clockwise orientation. This orientation is reversed for the internal boundary. The orientations of boundaries are reversible with the **Reverse/Delete Boundary** tool which will be described later. Next, for both circle and ellipse, specify the **Number of Elements**, which is the number of edges used to create the polygon. The default is 4 elements for creating a square. For **Center of Contour** specify x and y coordinates for the center of the contour separated by a comma. The default is 0,0. Click **Create** to confirm the boundary creation and close the window. Note that the contour geometry specified here can be modified using the **Modification** menu or **Edit Geometry** tool in the **Tool Bar**, which also will be described later.

Mesh Refinement: The refinement of the contour by splitting is performed by the **Refine Element** tool

in the **Tool Bar**. To initiate, click the tool button, then click the element to be divided. The user must be careful not to click the tip of an element, for it brings about another function which will be covered later. When a popup-menu appears, click the **Two Segments** menu item to have the element split into two equal segments. Repeat this procedure until the whole contour is refined. An alternative way is to click the **Multiple Segments** menu item when the popup-menu appears; enter the number of divisions you want to divide the specific element into, click **OK** to finish.

Nodal Coordinate Modification: To modify the coordinates of any node (internal, external or crack element), click the **Edit Geometry** tool icon in the **Tool Bar** and the cursor will turn to a cross hair. In the **Graphic Editor**, left click and hold the mouse to drag a node (a red dot) to move it around and see the new position of that node and the corresponding elements, release the mouse once you are satisfied with the geometry. To make fine adjustment of the location of the node, right-click the node and left-click the pop-up menu that appears as **Set Coordinates**. Type in the coordinates and click **Enter** to set the desired coordinate of that specific node. Note that, the right-click of the node for setting nodal coordinates only works when the cursor takes up a cross hair shape.

Double Node: To introduce double nodes, click **Refine Element** tool in the tool bar and the cursor will turn into a hand. Left-click a node to call a popup-menu with two menu items, **Double Node** and **Merge Elements**. Left-click **Double Node** to change that node into a double node. At each double node the nodal point is shifted a small distance from the end point of the element. It is also possible to revert a double node back to a single node by clicking a double node and selecting the **Single Node** menu item as long as the **Refine Element** tool is active. The same outcome of turning a node into a double or back to a single may be done through the **Modification Menu's Modify Boundary** command.

Merge Elements: Suppose you want to eliminate a node and merge the two adjacent element, then use the **Refine Element** tool in the **Tool Bar**. After setting the cursor to a hand, left-click a node to call the **Merge Elements** command. Click the menu item to create an element that replaces the previous two elements that meet at the node.

Merge Boundaries: Two boundaries can be merged together to form a composite shape. User should be careful and follow the instruction given in the **Information Bar** right above the **Graphic Editor** window, and carefully read the warnings (if any) that may show up. Click the **Merge Boundary** tool in the **Tool Bar** to initiate the process. Then click any one of the boundaries you would like to merge, and then click the second boundary to be merged with. The user must take precaution to ensure that the two merging boundaries have two and only two intersection points, any more than two would result in a warning message and the cancelation of the process. The orientations (clockwise or counter-clockwise) of the two boundaries are arbitrary and do not affect the orientation of the composite boundary. After the two boundaries have been chosen, all other unrelated boundaries will disappear for the rest of this merging process. Next, the user needs to determine which segments of the intersected boundary contours to keep to achieve the desired geometry. Once determined, click the two segments to keep. Notice that two segments need to be selected

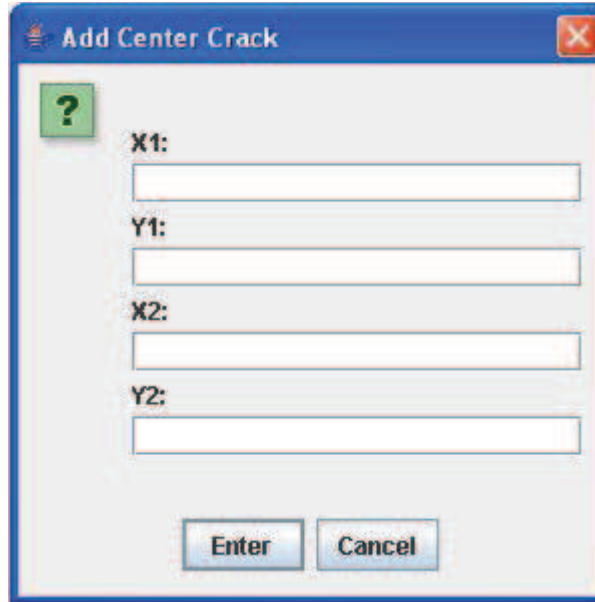


Figure 6: Add Center Crack window

among four possible segments. After two clicks, a dialog box will appear to ask for the orientation of the newly formed composite boundary. If anything goes wrong, then the process can be canceled anytime by right-clicking the mouse, the boundaries will be restored to the original state.

## 2.4 Center Crack

Straight Center Crack: Use the **Add Center Crack** tool in the **Tool Bar** to add a straight center crack. In the **Add Center Crack** window (Figure 6), specify the coordinates of the one end ( $x_1$  and  $y_1$ ) and the other end ( $x_2$  and  $y_2$ ) of the center crack. Note that all coordinates and lengths are non-dimensional. Click **Enter** to create the center crack. A red line in the window shows the newly created crack. Repeat the procedure for multiple cracks. This process introduces a single straight crack element for each crack.

Mesh Refinement: To refine the crack elements click the **Refine Element** tool in the **Tool Bar** first. Next, click a crack to refine in the **Graphic Editor** window. A popup-menu appears, click the **Two Segments** menu item to split the crack into two elements. Repeat this procedure for each crack element to split further into two equal elements. The progressive crack element refinement toward each crack tip is necessary for accurate results. The optimum crack element configuration is shown in Figure 7, where the size of the crack tip element is  $c = 1/16$  of the half crack length. To achieve this size, it is necessary to split the crack tip five times in a row: first to split the whole crack into two, second to split each of the two elements into two, third to split each of the two crack tip elements into two, and so on. Only the elements located at the crack tip need to be split. However, the use of a coarser crack tip element such as  $c = 1/4$ , which requires three splitting for each crack tip, gives results accurate enough for most applications. There is no merge

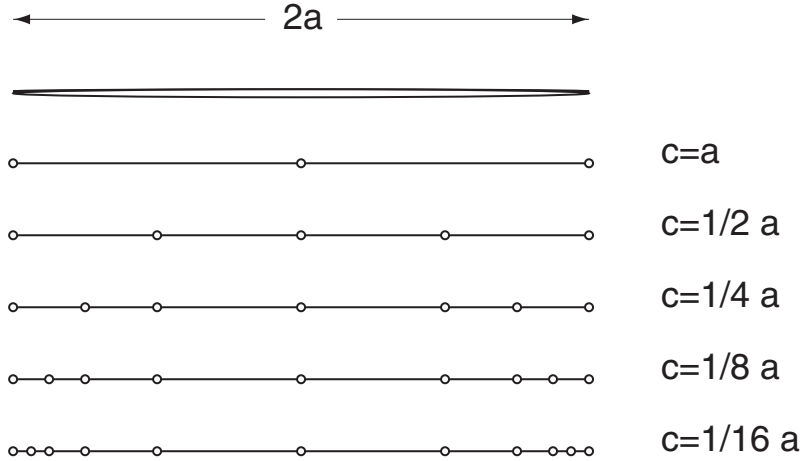


Figure 7: Crack elements for a center crack.

element option for the crack elements, if the you made a mistake, it can be corrected by the **Undo Modify Crack** command in the **Modification** menu bar, or simply delete the entire crack by the **Modify Crack** command, which is also under **Modification** menu bar.

Crack Shape Change: The method to change the size and shape of the crack is similar to that deployed for the boundary geometry change. Click the **Edit Geometry** tool in the **Tool Bar**, make sure the shape of the cursor is a cross hair. Left click and drag a crack node to move it to a new location. Right click on any crack node to specify the exact coordinate of the node. We can use the right-click technique to specify the coordinates of the nodes directly without click and drag. To introduce a kinked crack from a straight crack, this technique may be utilized. Note that the click-drag and right click works differently under different tools.

## 2.5 Edge Crack

Surface Breaking Point: The edge crack is introduced by the **Add Edge Crack** tool. After clicking the tool icon, a warning window opens up indicating that no double node is present; please create one. If double nodes are present, but they are not located for the prospective position of the surface breaking point of the crack, then it is necessary to introduce a node first and then turn it into a double node. If the (single) node is present at the surface breaking point, then simply turn it into a double node by **Refine Element** tool and right clicking the node. If the surface breaking point is not a node, then use the **Refine Element** tool to split the element into two to introduce a node at the surface breaking point and then make it a double node by right clicking the node.

Straight Edge Crack: Go back to click the **Add Edge Crack** tool. In the **Graphic Editor** window, click the double node at the surface breaking point of the edge crack. A window opens (Figure 8) requesting for the coordinates of the edge crack tip, which is the end of the crack away from the surface breaking point.

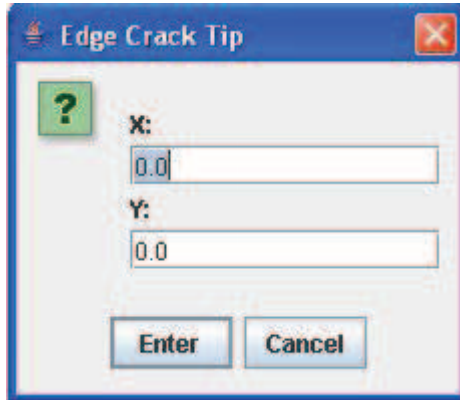


Figure 8: Add Edge Crack window

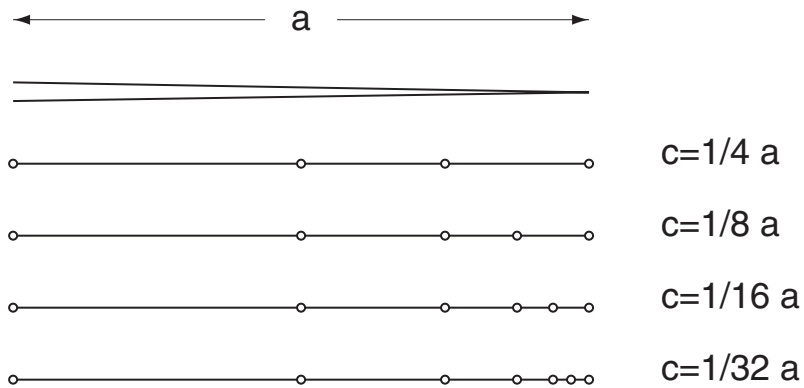


Figure 9: Crack elements for an edge crack.

Input the coordinates and click Enter. A red line is drawn from the surface breaking point to the crack tip to introduce the edge crack. The optimum crack element configuration for the straight edge crack is shown in (Figure 9), where the size of the crack tip element is again  $c = 1/16$  of the crack length. For the refinement and modification of the edge crack, follow the procedure described for the center crack.

## 2.6 Boundary conditions

General Strategy: In general, each node has three traction and three displacement components. Due to the displacement continuity, the end node of a continuous element has nine components: six traction (three independent for each element) and three displacements (three common for two elements). Any combinations of the traction and displacement boundary conditions that leave three unknown components are acceptable. For the double node, the situation is simpler since each node has total of six traction and displacement components and allows three unknown components. When the displacement components are specified at the double node, however, it is necessary to apply the same condition separately for each node.

Modifying Boundary Condition of a Single Element: Click **Refine Element** tool in the **Tool Bar** to initiate

the process. Then click on an element so a popup menu appears. Click the last menu item, **Modify Boundary Conditions** to open a window that says **Boundary Conditions for Element \***, where **\*** is the element number. The window has three sections separated into **First Node: node #**, **Middle Node** and **Last Node: node ##**, where **#** and **##** are the actual node numbers of the element. The left column of each section has three text-boxes for x, y and z nodal values of either the Force (or traction) or Displacement. The selection of either the Force or the Displacement boundary condition is made through the dropdown box in the right column. Select **Show Boundary Condition** checkbox in the lower left corner to display the boundary conditions specified using arrows for the traction and rollers for the displacement boundary conditions. The displacement boundary condition for the x and y components are shown by the number indicating the magnitude enclosed by a triangle, while a circle is used to indicate the z displacement component. Click **Accept** to accept the boundary condition and close the window.

Multiple Boundary Condition Selection: Quite often, multiple elements have the same boundary condition, so instead of specifying the boundary condition for each element, the user can select multiple elements and specify the boundary condition once. To carry out this task, click the **Refine Element** tool in the **Tool bar**. Left click and drag the mouse to draw a box that contains all the elements to be included. In order for a particular element to be included its both end nodes must be contained in the box. If only one node of an element is within the box, then it is not included in the list of elements. Once the desired elements are selected, simply release the mouse to allow the **Multiple Boundary Condition** dialog box to show. This dialog box is similar to that of the **Single Boundary Condition** dialog box except for the first section, which shows the element numbers that have been selected for this session of boundary condition modification. Once the desired boundaries are specified, click **Accept** to accept the changes and close the window. Also note that this feature can also be used to delete the boundary conditions. Simply select the elements, when the dialog box opens up, leave all the text-box values as the default value 0, click **Accept** to close the window.

Displacement Boundary Condition: Consider a continuous element  $E1$  for which the end node  $A$  of this element coincides with that of the adjacent element  $E2$ . If the displacement boundary condition is specified at the end node  $A$  of  $E1$ , it is not necessary to specify the same condition for node  $A$  of the element  $E2$ . Using the continuity condition, the displacement boundary condition of node  $A$  that belongs to the adjacent element  $E2$  is automatically set by the program. On the other hand, if the node is a double node, then the user must specify the same displacement boundary condition twice for the two adjacent elements that share the double node.

### 3 Observation points

If you want to calculate the displacement and traction in the post processing, click **Add Observation Point** tool in the **Tool Bar**. In the **Graphic Editor** window, click the location for an observation point. A window opens with the x and y coordinates of the clicked point. Modify if necessary. Input the angle of

the segment over which the traction components are calculated in degrees. This angle is not the angle of unit normal, rather it is the slope of the segment.

## 4 Create input file and Run the program

Save: To create the input file, either click **Save** in the **Tool Bar** or save through the **File** menu. A window opens to prompt for the name of the input file and the location to save. After saving the input file, a new window **Input: \*\*\*.inp** opens in the **Edit Window** which has now two window tabs, where **\*\*\*.inp** is the input file name. (To go back to the **Graphic Editor** window, click the tab at the top of the **Edit Window**.) Inspect the integrity of the input file. It is possible to edit the input file in this window. The edited input file is saved to replace the original input file created by the e-Handbook when the user clicks the **Run** tool.

Run: In order to run the program, click the **Run** icon in the **Tool Bar**. A window opens up to prompt for the name of the output file and the location to save it. After saving the output file, a window **Output: \*\*\*.out** opens in the **Edit** window which has now three window tabs, where **\*\*\*.out** stands for the output file name. The output files lists the boundary displacement and traction, stress intensity factors at the crack tips and the displacement and the traction at observation points.

Load: If you have an input file prepared manually or saved from other sessions, it is possible to load the file using the **Load** command in the **File** menu. This will load the contour geometry as well as the boundary conditions, observation points onto the **Graphic Editor** window. This is a great feature to test the integrity of the input file prepared manually.

## 5 Get stress intensity factors

After running the program, come back to the **Graphic Editor** window, click the **Crack Tip Info** tool which is the last icon in the **Tool Bar**. Click each crack tip and the stress intensity factors,  $K_I$ ,  $K_{II}$  and  $K_{III}$  will be listed in the **Information Bar** of the **Edit Window** located on the top of the window. Notice that K-values listed are real dimensional values dependent on the applied load and the characteristic length of the problem. For non-dimensional values, look for the output file in **Output: \*\*\*.out** window.