Engineering Software Research and Development, Inc.
10845 Olive Blvd., Suite 170
St. Louis, MO 63141-7760

Tech Brief

Crack Path Analysis

Authors: Dr. Ricardo Actis, Randy Dennison

Abstract

The objective of this work revolves around a need for an accurate numerical simulation method that computes the stress intensity factors (SIF) along a predicted crack propagation path for planar two-dimensional bodies. In response to this need, ESRD has developed an automated procedure in StressCheck®, a p-version finite element analysis (FEA) tool, that computes through-crack stress intensity factors for mode I and mode II along a predicted crack propagation path. This functionality is applicable to planar analysis (2D plate with in-plane loading). The trajectory of the crack is determined by a relationship between mode I and mode II SIF's (K_1 and K_2) and is shown to be very accurate.

1. Formulation

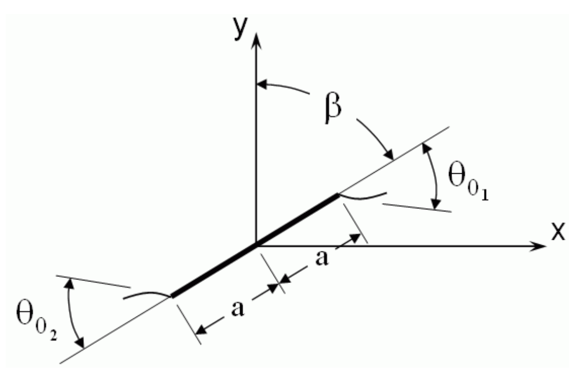
The formulation of the trajectory equation is based on two commonly recognized hypotheses [2] for the extension of cracks:

- The crack extension starts at its tip in a radial direction.
- The crack extension starts in the plane perpendicular to the direction of greatest tension.

These hypotheses mean that the crack will grow from the tip in the direction along which the tangential stress (σ_θ) is maximum and the shear stress ($\tau_{r\theta}$) is zero. In the general case of loading by mode I and II, the angle of crack extension, θ_0, is obtained from:

$$K_1 \sin\theta_0 + K_2(3\cos\theta_0 - 1) = 0 \qquad (1)$$

which gives the angle in terms of K_1 and K_2 measured from the tip of the crack, counterclockwise positive.



The solution of equation (1) is

$$\theta_0 = a\sin\left[K_2 \frac{K_1 \pm 3\sqrt{K_1^{\;2} + 8K_2^{\;2}}}{K_1^2 + 9K_2^2}\right] \tag{2}$$

For the special case of pure mode I loading ($K_2 = 0$),

$$\theta_0 = a\sin(0) = 0°$$

For pure mode II loading ($K_1 = 0$),

$$\theta_0 = a\sin\left(\pm\frac{\sqrt{8}}{3}\right) = \pm 70.5°$$

Both special cases are consistent with experimental observations.

For general loading both K_1 and K_2 may be different from zero, and equation (2) determines the trajectory of the crack. Reliable determination of the crack trajectory can be obtained only if the stress intensity factors, K_1 and K_2, are calculated accurately. For that reason StressCheck uses the Contour Integral Method (CIM) to compute the SIF's [1, 6].

2. Application of the method

To perform a *Crack Path* analysis, a "crack" object will be defined as part of the planar 2D model topology. The crack object has two tips, denoted "a" and "b". One or both tips can be set to "active" depending on its location within the model domain. The crack object has properties that are input by the user prior to runtime to control certain aspects of crack propagation, such as:

- a_{init} – Initial crack size (distance between tips "a" and "b")
- Δa – The crack increment dimension (enter separate values for "a" and "b")
- θ_{init} – Initial direction of crack growth for 1st increment (angle relative to global coordinates)
- a_{max} – The maximum crack length
- p-level – The polynomial order of the elements (for best results, set p=4 or 5)
- Run Limit – The maximum number of crack extension increments

These properties are entered in the Crack Path solution interface as shown in FIGURE 1 for each "active" crack. The value of the properties can be defined parametrically as was done for the initial

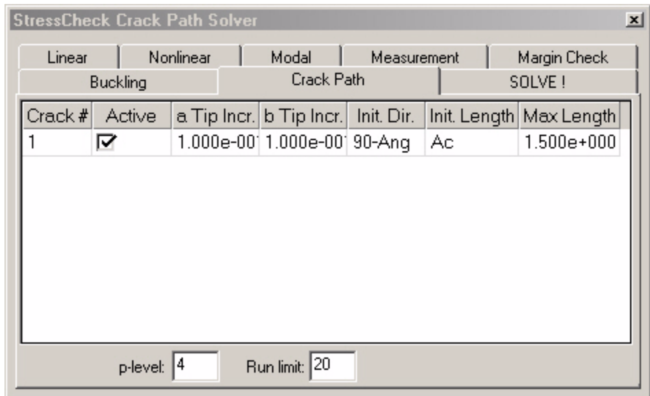


FIGURE 1 Solver Interface, Crack Path tab.

direction and initial length (FIGURE 1). This functionality enables the user to develop parametric Handbook models to support standardization and automation of analysis tasks.

Some important points to keep in mind:

- Careful consideration should be given to the value assigned to the crack increment (Δa). Specifying a crack increment that is too large will result in a crack path that is wavy. To obtain a smoother crack path, choose a smaller crack increment.
- When the crack gets close to an external boundary such that the contour integral radius goes outside the sheet body domain (FIGURE 2), the radius will be automatically reduced two more times in an effort to compute the stress intensity factors at that location. This is also true for cracks approaching regions of thickness change or a transition from one material to another.

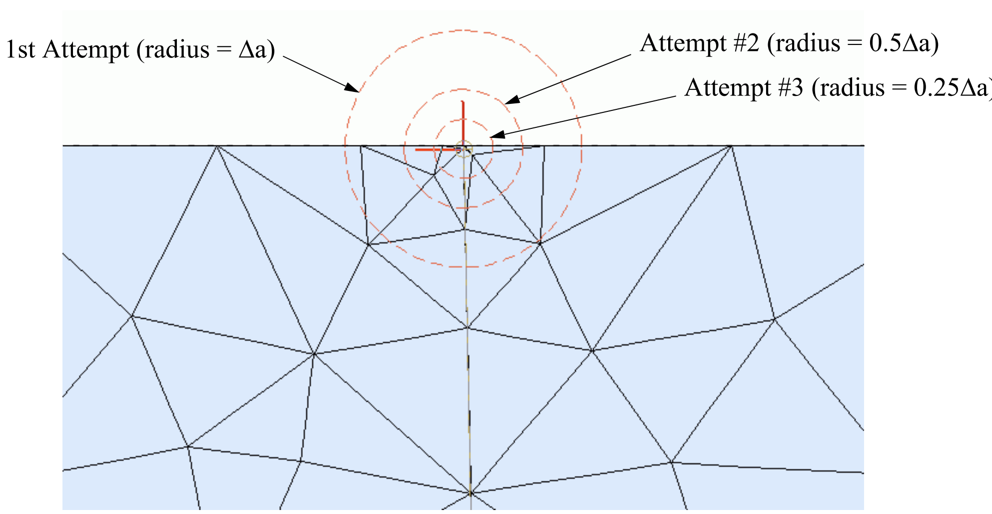


FIGURE 2 SIF Extraction. Contour Integral Path.

- When there are multiple cracks in the domain, all 'active' crack tips with non-zero increment values will grow simultaneously. As each crack reaches its specified maximum crack length (a^i_{max}), it will stop growing. The solution process will terminate when all cracks have reached their maximum or when the run limit is exceeded.
- The default solution Run Mode is set to *Automatic*. If you want to change the crack increment value(s) or make new cracks active during the analysis, then you must run in *Stepwise* mode. This is done within the SOLVE! tab in the Solver interface.
- The user can force a crack to propagate along a linear path with fixed orientation. This is done by defining a special parameter "_fix_crack_angle" with a value set equal to the orientation angle (in degrees) relative to the global coordinate system.

The model mesh is constructed by an automatic meshing routine that recognizes the crack object and generates a refined mesh at the crack tip. Because the mesh and geometric object numbers change for each crack increment, the user must adhere to certain model construction rules as noted below:

- The domain (planar body) must be a single trimmed sheet body and the crack object must lie in the domain.
- The user must declare an initial crack length. The length can be zero, meaning that StressCheck will determine the orientation of the crack during the first solution and automatically add an initial crack during the second iteration.
- The mesh density of the global model is defined in the automesh record. Default automesh settings are recommended, however, the user can control certain aspects of the global meshing behavior by changing mesh attributes (D/H, Ratio, MinLen, Trans). See Users Guide for more details. If the user does not supply this record, it will be created at runtime using default values.
- Assignment of thickness, material properties, boundary conditions, and p-discretization must be described using "All" records or "region" sets. The reason for this is because the numbering of geometric and mesh objects may change during the solution process. A *region* set is a set of objects that lie within a specified region of the model and an *All* set means that every object of the specified type is selected.

There are essentially three different cases that can be considered for Crack Path analysis:

1) The location of the initial crack is known and one tip lies on an edge of the sheet body. Thus, there is a single "active" crack tip.

2) The location of the initial crack is known and lies internal to the sheet body. In this case the crack has two "active" tips from which to propagate.

3) The location of the initial crack is not known explicitely but is assumed to initiate somewhere on a specified boundary (e.g. hole wall or edge of part).

Examples of each of these cases will be described in the following sections.

Example 1: Three-Point Bend Test Specimen

Consider a 0.5 inch thick steel plate (E = 30e6 psi, ν = 0.3, plane-stress), loaded in 3-point bending as shown in FIGURE 3. The plate represents a test specimen which has an initial through crack of length (a_{init} = 1.5 inch) on the lower surface five inches from the left-hand corner. A row of three holes is located along a vertical axis six inches from the left-hand edge of the plate. This first example illustrates a typical case when the initial crack location is known and the user is interested in simulating crack propagation from a single crack tip.

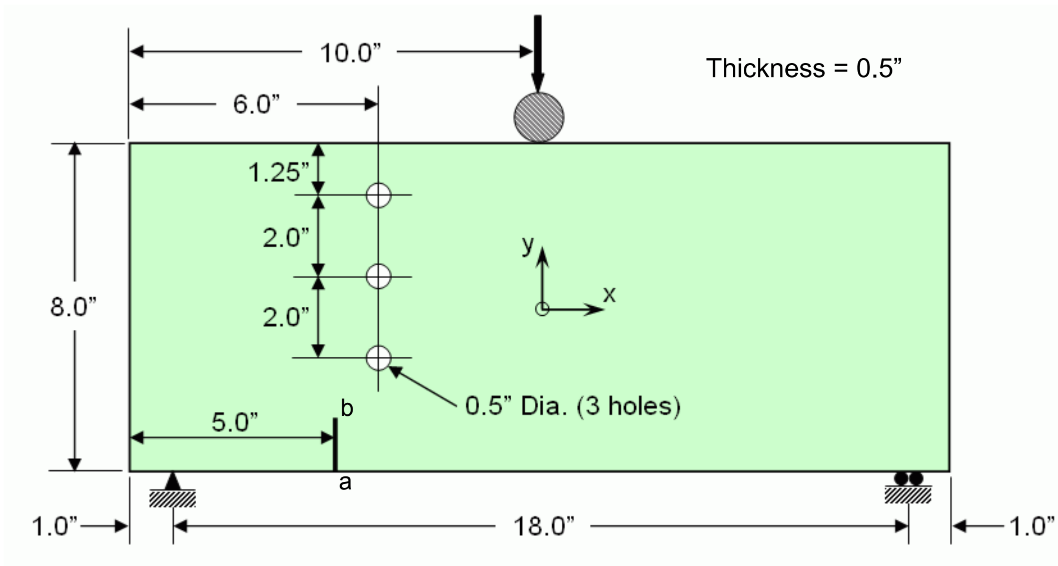


FIGURE 3 Problem Description for Example 1.

To place the *Crack* object in the model, select the Create Model icon and complete the following:

- Mesh tab > Create > Crack > Locate > Input on > X: -5.0 > Y: -4.0 > Z: 0.0 > Length: 1.5 > Angle: 90 > Accept.

Note that the origin of the crack object is at the a-Tip and the crack is oriented 90 degrees relative to the global coordinate system.

After defining the model domain, locating the crack object, and assigning attributes, you are ready to initiate the crack path analysis. Go to the Solver interface and select the Crack Path tab. In this interface you will see a record of the crack object with the default status set to not active. To include a crack in the analysis, check the *Active* box next to it. Next, supply the crack increment (0.0 for a-Tip and 0.25 for b-Tip) to be added at the end of the crack after each iteration and all other information as shown in FIGURE 4 (Init. Dir.: 90, Init. Length: 1.5, Max. Length: 5.0, p-level: 4, Run limit: 15).

The polynomial order does not need to be high (p=4 in this case) because the mesh generated during the crack path analysis is quite dense. The user can run a second analysis with p=5 to verify convergence of the stress intensities and compare the crack trajectory.

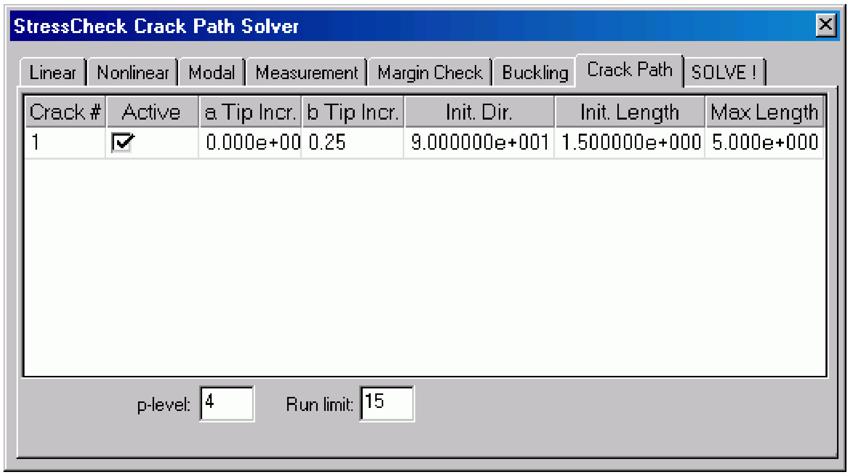


FIGURE 4 Crack Path Solver Interface for Example 1.

Next, choose the SOLVE! tab to invoke the solution process. Complete the following information:

- SOLVE! tab > Initialize > Automatic > Iterative > Click the Solve button.

The solution window in the solver execution interface displays the status of the execution. The status bar located at the bottom of the main window displays which operation is being performed for each run. Opon completion of the analysis (increment reaches the maximum crack size or run limit, or the crack grows to the edge of the model domain), a summary of the crack path analysis will be displayed in the Edit window as shown in FIGURE 5. In this example the Run Limit of fifteen and maximum crack length of 5 inches is reached simultaneously and stops the analysis.

To perform post-processing operations, select the View Results icon  from the Main Toolbar. To display the deformed shape as shown in FIGURE 6, select the Plot tab and complete the required information as follows:

- Plot tab > Select > All Elements > Selection > Solution: SOL > Run: 1 > Shape: Deform > Click the Plot button.

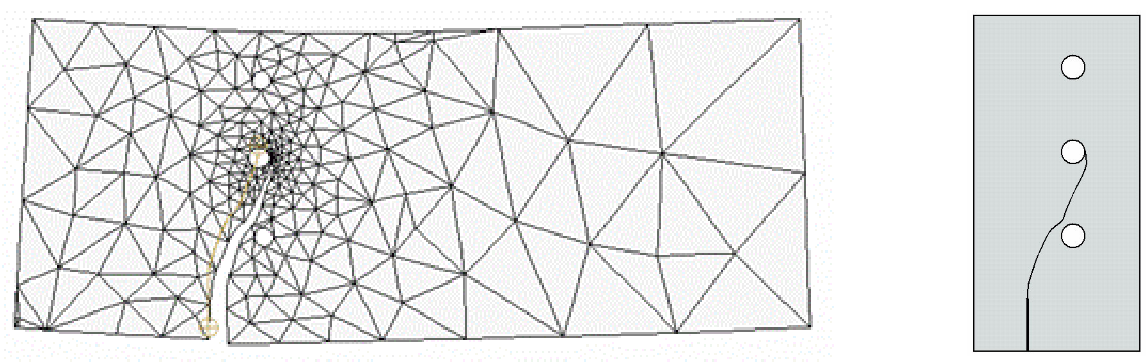
Post-processing operations are available for the last run only, that is, for the final crack length. Comparison of the predicted crack trajectory to experimental test [5] demonstrates the robustness of the crack path algorithm.

StressCheck Edit 1 [j1.rtf]

Crack Path Summary
Bending Specimen

Run #	Crack	Tip	X	Y	K1	K2	Angle	Length
1	1	1	-5.0000e+000	-2.5000e+000	1.1369e+000	1.9279e-001	71.73	1.5000e+000
2	1	1	-4.9216e+000	-2.2626e+000	1.3151e+000	-7.9229e-002	72.22	1.7516e+000
3	1	1	-4.8453e+000	-2.0245e+000	1.4783e+000	5.2571e-002	70.11	2.0014e+000
4	1	1	-4.7602e+000	-1.7895e+000	1.6698e+000	7.6230e-004	68.99	2.2513e+000
5	1	1	-4.6706e+000	-1.5561e+000	1.8942e+000	5.2904e-002	65.80	2.5013e+000
6	1	1	-4.5681e+000	-1.3280e+000	2.1271e+000	1.1390e-001	58.93	2.7513e+000
7	1	1	-4.4391e+000	-1.1139e+000	2.1393e+000	1.8057e-001	47.89	3.0015e+000
8	1	1	-4.2714e+000	-9.2846e-001	1.8120e+000	-1.3374e-001	53.79	3.2519e+000
9	1	1	-4.1237e+000	-7.2675e-001	2.2622e+000	-3.1638e-001	71.29	3.5020e+000
10	1	1	-4.0436e+000	-4.8996e-001	2.7128e+000	1.5462e-001	68.47	3.7532e+000
11	1	1	-3.9518e+000	-2.5740e-001	3.0829e+000	-2.5716e-002	66.80	4.0032e+000
12	1	1	-3.8533e+000	-2.7625e-002	3.5075e+000	-6.0523e-002	68.82	4.2532e+000
13	1	1	-3.7630e+000	2.0549e-001	4.0721e+000	-1.4530e-001	73.36	4.5032e+000
14	1	1	-3.6914e+000	4.4501e-001	4.6618e+000	-5.3344e-001	87.03	4.7533e+000
15	1	1	-3.6784e+000	6.9468e-001	4.0691e+000	-8.2141e-001	111.32	5.0039e+000

FIGURE 5 Crack Path Analysis Summary for Example 1.



Predicted   Experimental[4,5]

FIGURE 6 Deformed Shape and Crack Trajectory for Example 1.

Example 2: Plate with Internal Crack

Consider the rectangular panel shown in FIGURE 7 loaded by a uniform traction T_y=30000 psi at the upper end and fixed at the base. The panel is of unit thickness and has a height to width ratio of 4/1. The material is 6Al-4V titanium (E = 16e6 psi, ν = 0.29, plane-stress). An initial internal crack of length (a_{init} = 0.56 inch) is centered in the panel and oriented 30 degrees from the vertical (y-axis).

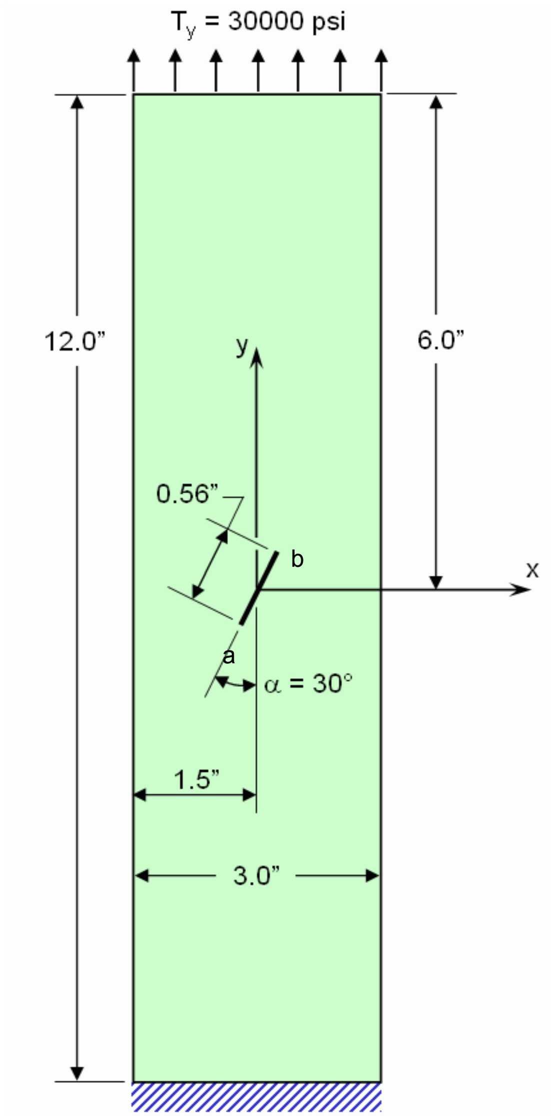


FIGURE 7 Problem Description for Example 2.

Before creating the solution domain, define two parameters that can be used to control the length and orientation of the initial crack. Select the Model Info icon from the Main Toolbar and select the Parameters tab. Enter the following information:

- Name: Ac > Description: Total crack length > Value: 0.56 > Accept
- Name: Ang > Description: Crack orientation from vertical (degrees) > Value: 30 > Accept

As was done in the previous example, the crack object will be placed in the model domain using the *Locate* method where the a-Tip is the origin of the crack object. Select the Create Model icon and complete the following:

- Mesh tab > Create > Crack > Locate > Input on > X: -Ac/2*cos(rad(90-Ang)) > Y: -Ac/2*sin(rad(90-Ang)) > Z: 0.0 > Length: Ac > Angle: 90-Ang > Accept.

After defining the model domain, attaching the crack object, and assigning attributes, go to the Solver interface and select the Crack Path tab. Check the *Active* box next to Crack #1 and supply the following information:

(a Tip Incr.: 0.05, b Tip Incr.: 0.05, Init. Dir.: 90-Ang, Init. Length: Ac, Max. Length: 1.5, p-level: 4, Run limit: 20).

The results of the analysis are shown in FIGURE 8 and FIGURE 9.

StressCheck 1 Edit [j7.rtf]

Crack Path Summary
Panel in Tension

Run #	Crack	Tip	X	Y	K1	K2	Angle	Length
1	1	2	-1.4000e-001	-2.4249e-001	7.2882e+003	1.2390e+004	180.19	5.6000e-001
1	1	1	1.4000e-001	2.4249e-001	7.2935e+003	1.2387e+004	0.20	5.6000e-001
2	1	2	-1.9000e-001	-2.4265e-001	2.5155e+004	-1.5670e+003	173.45	6.6436e-001
2	1	1	1.9000e-001	2.4266e-001	2.5147e+004	-1.5713e+003	353.46	6.6436e-001
3	1	2	-2.3967e-001	-2.3695e-001	2.8502e+004	1.1008e+003	171.33	7.6395e-001
3	1	1	2.3967e-001	2.3697e-001	2.8505e+004	1.1677e+003	351.28	7.6395e-001
4	1	2	-2.8910e-001	-2.2941e-001	3.1332e+004	-1.0879e+003	174.16	8.6388e-001
4	1	1	2.8910e-001	2.2939e-001	3.1337e+004	-1.1082e+003	354.21	8.6388e-001
5	1	2	-3.3884e-001	-2.2433e-001	3.4013e+004	2.6054e+002	175.13	9.6387e-001
5	1	1	3.3884e-001	2.2434e-001	3.4014e+004	3.3597e+002	355.31	9.6387e-001
6	1	2	-3.8866e-001	-2.2008e-001	3.6542e+004	-6.4789e+001	175.10	1.0639e+000
6	1	1	3.8867e-001	2.2025e-001	3.6564e+004	6.8281e+001	355.31	1.0639e+000
7	1	2	-4.3848e-001	-2.1582e-001	3.9092e+004	-3.5000e+002	175.11	1.1639e+000
7	1	1	4.3851e-001	2.1617e-001	3.9057e+004	-3.3118e+002	355.31	1.1639e+000
8	1	2	-4.8830e-001	-2.1155e-001	4.1550e+004	-7.1776e+002	177.08	1.2639e+000
8	1	1	4.8834e-001	2.1208e-001	4.1553e+004	-6.2038e+002	357.02	1.2639e+000
9	1	2	-5.3823e-001	-2.0901e-001	4.4202e+004	1.1476e+002	177.57	1.3639e+000
9	1	1	5.3827e-001	2.0948e-001	4.4205e+004	8.9039e+001	357.45	1.3639e+000
10	1	2	-5.8819e-001	-2.0689e-001	4.6864e+004	-6.8990e+001	177.56	1.4639e+000
10	1	1	5.8822e-001	2.0726e-001	4.6861e+004	-1.4072e+002	357.45	1.4639e+000
11	1	2	-6.3814e-001	-2.0476e-001	4.9580e+004	-3.6600e+002	177.56	1.5639e+000
11	1	1	6.3817e-001	2.0503e-001	4.9618e+004	-3.4188e+002	357.45	1.5639e+000

FIGURE 8 Crack Path Analysis Summary for Example 2.

Notice that both crack tips increment by 0.05 inch for each run. The user can specify different crack propagation increments for each tip.

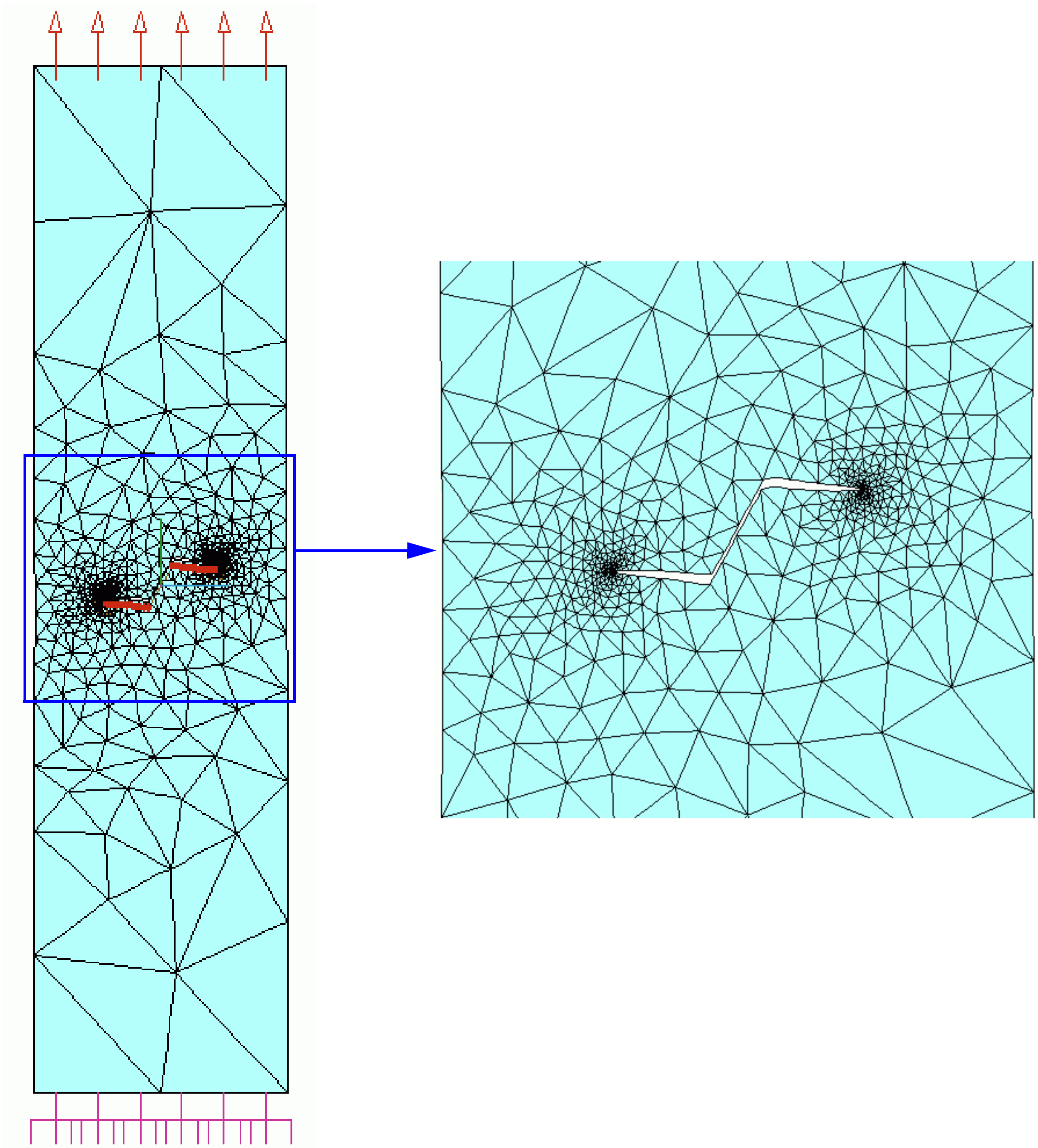


FIGURE 9 Deformed Shape and Crack Trajectory for Example 2.

Example 3: Nut-Plate Holes in a Uniform Stress Field

Consider a rectangular plate of unit thickness with a one inch diameter fastener hole located in the center of the plate and two smaller holes used to install a Nut-plate fastener system as shown in FIGURE 10. The Nut-plate is oriented 45 degrees relative to the global x-axis of the plate. The left edge of the plate is fixed and the right edge is loaded by a uniform normal traction (T_x) and a shear traction (T_y). The material is 7075-T6 aluminum (E = 10.5e6 psi, ν = 0.31, plane-stress).

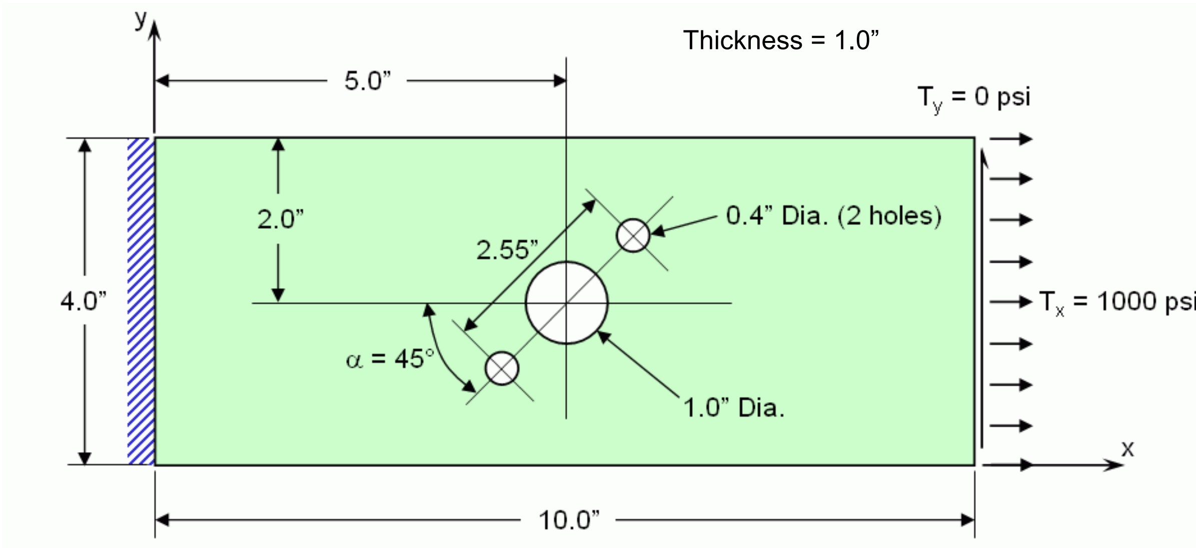


FIGURE 10 Problem Description for Example 3.

One can imagine that this model is defined parametrically such that the user can change the geometric dimensions, the material properties, or the magnitude and direction of the applied tractions to account for a multitude of possible variations. The problem is that the critical location ($K_t\sigma_{max}$) will change as the problem definition changes.

In this example we are interested in obtaining the stress intensity factors (K_1 and K_2) along a path that originates on the larger ‘center’ hole for two cases; 1) T_x = 1000, T_y = 0, and 2) T_x = 0, T_y = -1000. As before, we create the solution domain (sheet body) and apply material properties, thickness and boundary conditions using ‘Region’ sets or ‘All’ records. Since we do not know where the crack will initiate, we use the *Auto* method to attach the crack object to a circle that is not part of the sheet body but occupies the same space as the center hole (FIGURE 11).

- Mesh tab > Create > Crack > Auto. Leave all input fields off, and with the mouse select the circle that is not part of the sheet body. A symbol ⊕ representing the Auto Crack object will be shown on the model.

After defining the model domain, attaching the crack object, and assigning attributes, go to the Solver interface  and select the Crack Path tab. Check the *Active* box next to Crack #1 and supply the fol-

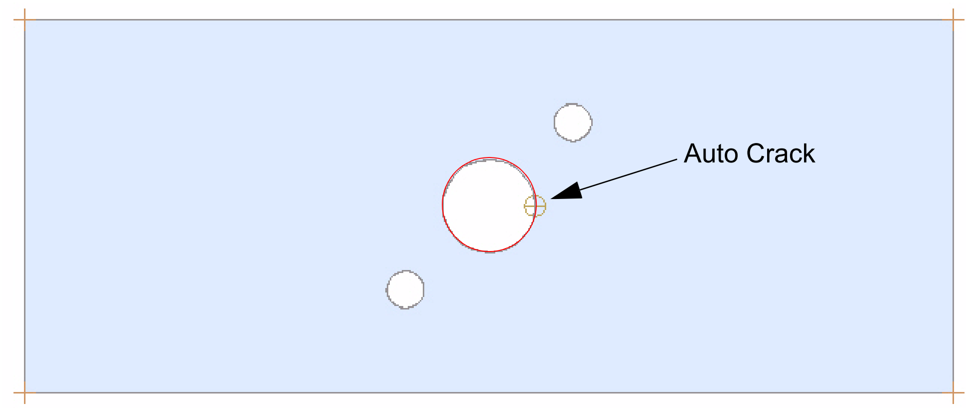


FIGURE 11 Attaching the Auto Crack to a Geometric Object.

lowing information as shown in FIGURE 12 (a Tip Incr.: 0.1, b Tip Incr.: 0.1, Init. Dir.: 0, Init. Length: 0, Max. Length: 1.5, p-level: 5, Run limit: 15).

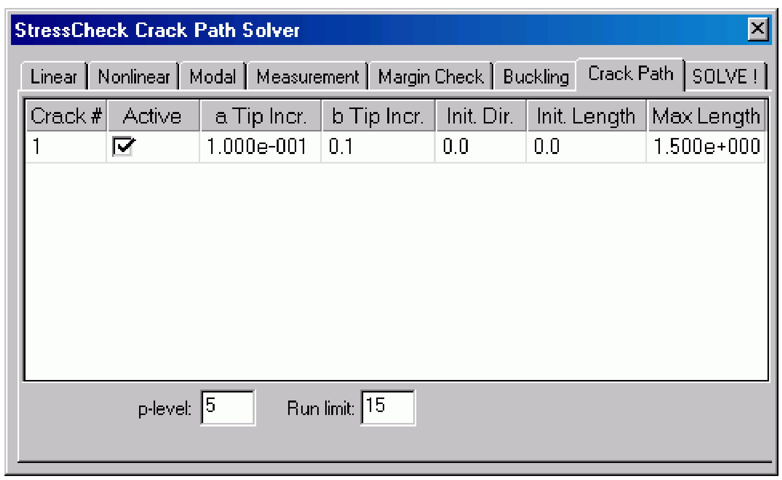


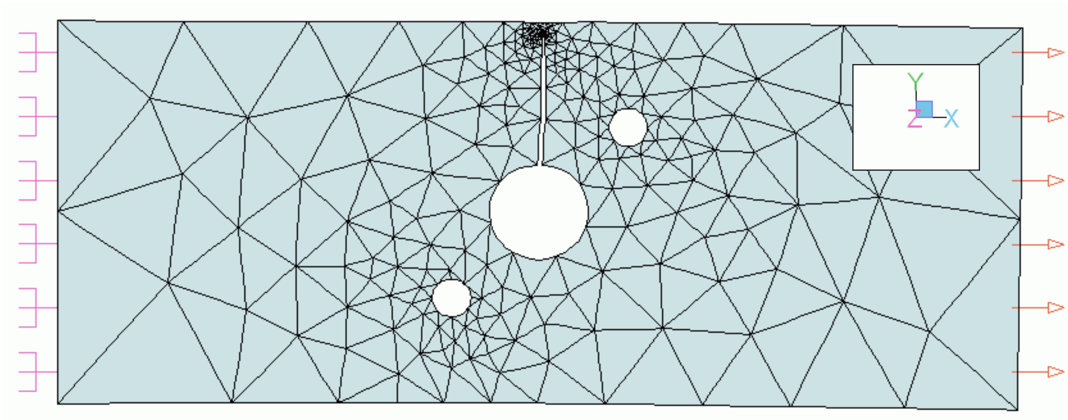
FIGURE 12 Crack Path Solver Interface for Example 3.

Notice that the crack increment for tips “a” and “b” were both assigned the same non-zero value and yet the crack will only propagate from one of these tips. The reason for this is explained in the following:

1) Run #1 is a linear analysis of the “unflawed” geometry. From this solution, the maximum first principal stress is computed at discrete points along the geometric object (in this case the circle) to which the Auto Crack was attached. The location of the maximum stress is recorded along with the principal direction.

2) A test is made for crack tip “a”. If the increment and computed direction places the new crack tip inside the model domain, then the Crack Path analysis initiates (Run #2) using crack tip “a”. Otherwise, crack tip “b” is used in the analysis.

The results of the two analyses are shown below in FIGURE 13 and FIGURE 14.



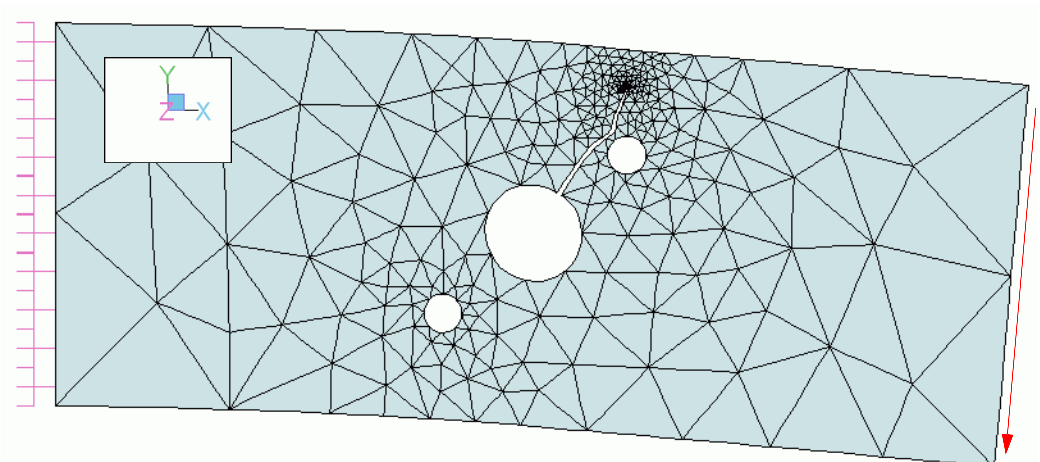
Deformed Shape and Crack Path

Crack Path Summary
Orientated Nut-Plate (3 Holes)

Run #	Crack	Tip	X	Y	K1	K2	Angle	Length
2	1	1	9.4653e-003	5.9993e-001	1.5090e+003	3.0890e+001	86.69	1.0000e-001
3	1	1	1.5242e-002	6.9976e-001	1.6934e+003	1.2242e+000	85.70	2.0002e-001
4	1	1	2.2747e-002	7.9948e-001	1.7610e+003	4.4478e+000	85.73	3.0001e-001
5	1	1	3.0200e-002	8.9920e-001	1.8060e+003	-7.0417e+000	85.72	4.0002e-001
6	1	1	3.7657e-002	9.9892e-001	1.8601e+003	-3.3103e+001	87.76	5.0002e-001
7	1	1	4.1563e-002	1.0988e+000	1.9412e+003	-1.5288e+001	88.20	6.0002e-001
8	1	1	4.4705e-002	1.1988e+000	2.0517e+003	-4.4426e+001	90.65	7.0002e-001
9	1	1	4.3568e-002	1.2988e+000	2.1997e+003	7.5848e+000	91.22	8.0003e-001
10	1	1	4.1441e-002	1.3988e+000	2.3848e+003	1.4707e+000	91.20	9.0003e-001
11	1	1	3.9354e-002	1.4987e+000	2.6206e+003	1.2488e-001	91.20	1.0000e+000
12	1	1	3.7266e-002	1.5987e+000	2.9337e+003	2.4623e+000	91.20	1.1000e+000
13	1	1	3.5178e-002	1.6987e+000	3.3830e+003	8.8343e+000	91.20	1.2000e+000
14	1	1	3.3090e-002	1.7987e+000	4.1253e+003	2.1262e+001	91.20	1.3000e+000
15	1	1	3.1002e-002	1.8987e+000	5.7566e+003	5.3132e+001	91.20	1.4000e+000

Tabulated Results

FIGURE 13 Results for Example 3. Load Case #1.



Deformed Shape and Crack Path

StressCheck Edit 1 [j1.rtf]

Crack Path Summary
Perforated Plate (3 holes)

Run #	Crack	Tip	X	Y	K1	K2	Angle	Length
2	1	1	2.9444e-001	5.2279e-001	3.7175e+003	3.4917e+000	60.70	1.0000e-001
3	1	1	3.4338e-001	6.0999e-001	4.3859e+003	3.0913e+001	60.70	2.0000e-001
4	1	1	3.9232e-001	6.9720e-001	4.8884e+003	9.5999e+001	58.45	3.0000e-001
5	1	1	4.4465e-001	7.8241e-001	5.3607e+003	5.6568e+001	56.72	4.0001e-001
6	1	1	4.9952e-001	8.6601e-001	5.7381e+003	2.0824e+002	52.29	5.0001e-001
7	1	1	5.6068e-001	9.4513e-001	5.7983e+003	2.2991e+002	46.75	6.0004e-001
8	1	1	6.2920e-001	1.0180e+000	5.1210e+003	1.8943e+002	41.39	7.0008e-001
9	1	1	7.0423e-001	1.0841e+000	3.6243e+003	-4.6179e+002	54.51	8.0011e-001
10	1	1	7.6228e-001	1.1655e+000	4.6282e+003	-1.3125e+003	85.57	9.0037e-001
11	1	1	7.7001e-001	1.2652e+000	6.5287e+003	1.2371e+003	71.76	1.0017e+000
12	1	1	8.0131e-001	1.3602e+000	8.0516e+003	-5.0348e+002	74.48	1.1019e+000
13	1	1	8.2807e-001	1.4565e+000	9.1981e+003	4.1092e+002	71.44	1.2019e+000
14	1	1	8.5990e-001	1.5513e+000	1.0444e+004	-4.0160e+002	74.61	1.3018e+000
15	1	1	8.8643e-001	1.6478e+000	1.2056e+004	1.2417e+002	74.57	1.4018e+000

Tabulated Results

FIGURE 14 Results for Example 3. Load Case #2.

References

[1] B. A. Szabó and I. Babuska. *Finite Element Analysis,* John Wiley and Sons, Inc. New York (1991).

[2] F. Erdogan, G. C. Sih. “On the Crack Extension in Plates Under Plane Loading and Transverse Shear”, *Journal of Basic Engineering*, 519-527, (Dec. 1963).

[3] B. N. Rao, S. Rahman, “An Efficient Meshless Method for Fracture Analysis of Cracks”, *Computational Mechanics 26*, (2000) 398-408. Examples 4 & 5.

[4] Carvalho, et. al., “Automatic Fatigue Crack Propagation Using a Self-Adaptive Strategy”.

[5] A. R. Ingraffea, M. Grigoriu, “Probabilistic Fracture Mechanics: A Validation of Predictive Capability”, An Interim Report for AFOSR Project FQ8671-8900185, Report 89-11, Cornell University, (Nov. 1989).

[6] D. Broek, “The Practical Use of Fracture Mechanics”, Kluwer Academic Publishers, The Netherlands, (1989).