International Conference on Economic Engineering and Manufacturing Systems
Braşov, 26 – 27 November 2009

RELIABILITY OF THE FEM CALCULATIONS OF THE FRACTURE MECHANICS PARAMETERS

Galina TODOROVA, Valentin DIKOV
Technical University of Sofia, Bulgaria

Abstract. This paper describes some results obtained by analyzing cracked plates by means of FEM based on the procedures of ANSYS 10. The numerical results are compared with their analytical solution. The influence of mesh configurations on crack characteristic values is considered. It is found that the simple isoparametric elements are not appropriate for calculating SIF and their usage has to be avoided. ANSYS gives fictitious results about K_{II} and they must be neglected.

Keywords: linear fracture mechanics parameters, Stress Intensity Factors (SIF), FEM, ANSYS

1. Introduction

The modeling of problems in the field of linear mechanics of destruction (LMD) by means of isoparametric quadrilateral finite elements of the finite element method (FEM) creates some difficulties. The use of polynomial functions for such purposes does not permit correct presentation of the singular areas of the stresses and deformations at the crack tip. So during the size reduction of the finite elements (i.e. a finer grid) the solution obtained by FEM initially begins to converge to the theoretical one until some critical degree. After that a deviation from this trend is observed. This particularity has been showed [1] in the earlier development of FEM. Several researchers have proposed singular interpolation functions [2, 3 ,4]. These finite elements with special functions are not available in standard programs for FEM calculations. Considerable success in the use of FEM for solving tasks in the field of LDM is the development of "quarter-point" (¼) of the final element by Henshell, Shaw and Barsoum [5, 6]. They proved that the correct fields of displacements, stresses and deformations at the crack tip could be numerically modeled (for linear environment) by moving the node (position ½) to the crack tip (position ¼). This procedure introduces singularities in the stress field.

The determination of linear fracture mechanics parameters – Stress Intensity Factors (SIF) plays an important role in fracture analysis. The brittle failure state of structures could be estimated by comparing these parameters with their particular critical values.

That is why the aim of this paper is to establish the reliability of the numerical solution for Stress Intensity Factor using ANSYS. In order to compare the results from ANSYS a simple geometry is chosen because of the availability of its analytical solution in literature.

2. Experimental part

2.1. The model

Specimen geometry: (figure 1): Plate with central crack with $2a=2.10^{-2}$ m length. The plate dimensions are: $H = 10.10^{-2}$, $W = 20.10^{-2}$, $t = 0,6.10^{-2}$ m.

Crack: A crack is placed perpendicularly to the loading direction in the center of the plate. The center-cracked tension plate is assumed to be in the plane stress condition in the present analysis.

Material model: linear elastic isotropic with modulus of elasticity E = 0,86 GPa and Poisson ratio $\nu = 0.3$.

Boundary conditions: The elastic plate is subjected to an uniform tensile stress in the longitudinal direction as much as σ= 3135 Pa.

The calculation procedure presented below is an analytical solution for centre cracked plates under tension (figure 1). According to Murakami [7] SIF K_I (for a finite plate, plane stress is assumed) is defined as:

$$K_I = \sigma\sqrt{\pi a}F_I(\alpha,\beta) \quad (1)$$

where $F_I(\alpha,\beta)$ is a function of geometry of the plate.

Coefficients α and β are defined as follows:

$$\alpha = \frac{2a}{W} \quad (2)$$

$$\beta = \frac{2H}{W}; \qquad (3)$$

For the plate chosen:
$2a = 2.10^{-2}$ m, and $2H = W = 20.\ 10^{-2}$ m.

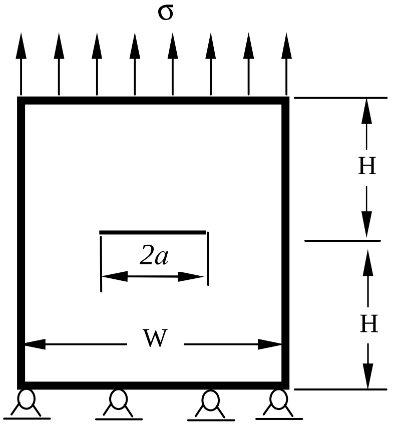

Figure 1. A scheme of the experimental state and plate parameters used for analytical solution

For numerical solution a rectangular isoparametric element which is familiar as “Quad 8-node 82” could be used. It has nodes at the corners and also at the midpoint on its each side. Another similar element (Fig.2) has midpoints which are moved one-quarter side distance to the node - placed at the crack tip position. Such a ¼ point element is also called “a singular element”.

The displacement extrapolation method is applied, using a fit of the nodal displacements in the vicinity of the crack, according to the 1/(r)½ approximation.

The elements designed for numerical computation of SIF in two-dimensional space are ¼ point elements: a quadrilateral ¼ point element (figure 2) and a collapsed ¼ point element (figure 3).

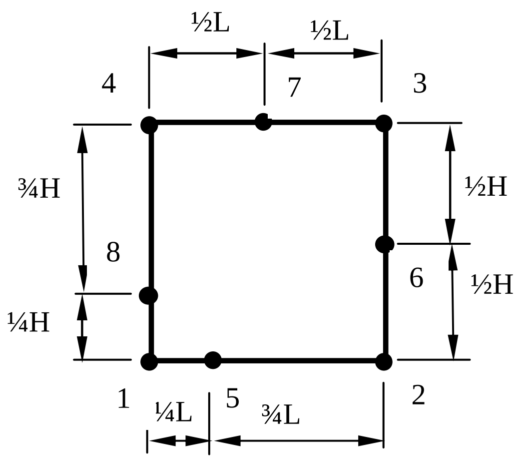

Figure 2. A conception of a quadrilateral ¼ point (singular) element in the two-dimensional space

The quadrilateral ¼ point element is not implemented in ANSYS.

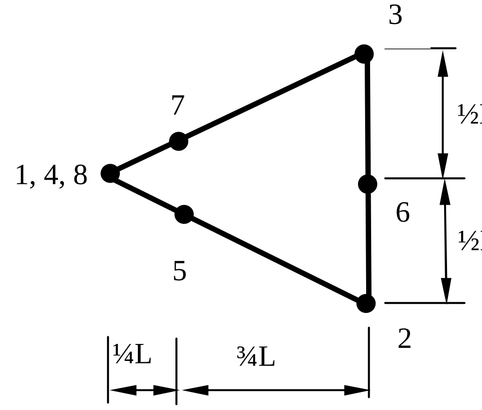

Figure 3. A conception of a collapsed (triangular) ¼ point (singular) element in the two-dimensional space

Type displacement, stress and strain fields are modeled by moving of the mid-side node of the element to the position of ¼ nearer to the crack tip – as it is shown on Fig.2./points 5 and 8/ and on figure 3 /points 5 and 7/.

ANSYS software is equipped only with a 2D triangular singular element, but neither with 2D rectangular nor with 3D singular elements.

Around the node at the crack tip, a circular area is created and it is divided into a designated number of triangular singular elements. These are precisely the elements which can interpolate the stress distribution in the vicinity of the crack tip. They introduce 1/√r singularity where r is the distance from the crack tip (r/a<<1).

These elements are implemented for FEM calculations of the SIF – KI.

2.2. Numerical simulation and calculation of fracture mechanics parameters for cracked plates

The computation of SIF is performed using ANSYS software. The general steps are:

- Input of the material, geometry, the boundary and loading conditions for FEM analysis of the plate.

- The crack points, the length of the crack, the crack position and the sides of the crack are input data.

- Meshing the structures. The local FE mesh is refined at the crack tip.

- Using operators KCALC for linear problems to obtain SIF.

3. Results and discussion

3.1. Analytical solution

The data presented above allow us to calculate the values α and β:

$$\alpha = \frac{2a}{W} = \frac{2.10^{-2}}{20.10^{-2}} = \frac{1}{10};$$

$$\beta = \frac{2H}{W} = \frac{20.10^{-2}}{20.10^{-2}} = 1;$$

Now it is easy to obtain the analytical value for K_I:

$$K_I = \sigma\sqrt{\pi a}F_I(\alpha,\beta) =$$
$$= 3135.\sqrt{\pi.1.10^{-2}}.1{,}014 =$$
$$= 563{,}30\ Pa\sqrt{m}$$

This K_I value will be used as reference to the results obtained from FEM method.

3.2. Numerical solution using ANSYS

The results from a numerical investigation on the dependence of the mesh size on the fracture mechanics parameters are presented. They refer to plates with transversal to the loading force cracks.

Mesh №1

The mesh (Fig.4) is generated according to [8]. To obtain reliable results ¼ finite elements are used. The radius of the first row of elements, generated in the vicinity of the crack tip, is:,

$$r = \frac{2a}{8} = \frac{2.10^{-2}}{8} = 0{,}0025m$$

where 2a is the length of the crack. Every one of these elements is positioned at 30° or 40° in the circumferential direction. The mesh generated contains 14416 elements and 29784 nodes (Fig.5). Full model is used regardless of the symmetry of geometric shape and loading of the plate.

SIF are calculated by nodal results during post-processing after definition of the path along the crack face. The numerical results are shown in Listing 1.

$$\Delta K_{I_1} = \frac{K_I - K_{I\ numerical\ value}}{K_I}\cdot 100 =$$
$$= \frac{563.30 - 563.07}{563.30}\cdot 100 = 0.04\%$$

Note that K_I = 563.30 is the referent value from the analytical solution!

Mesh №2

The mesh (figure 6) is generated using double reduction of the radius of the first row of elements generated near the crack tip as follows:

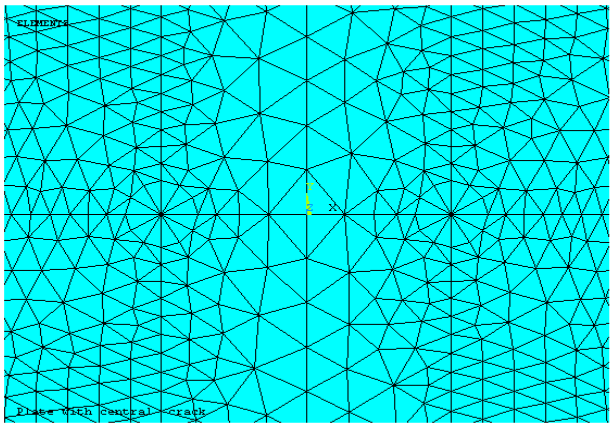

Figure 4. Mesh, r = 2a/8

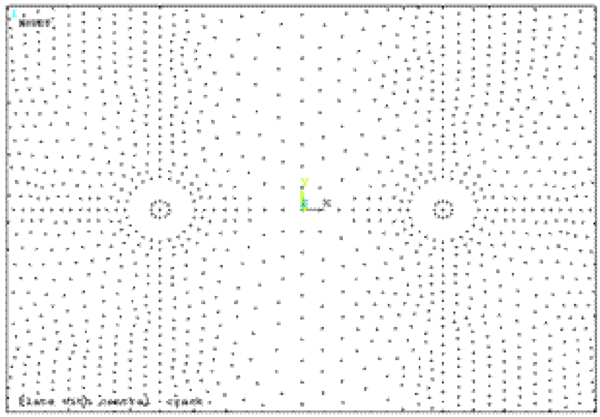

Figure 5. Nodes of the mesh, r = 2a/8

$$r = \frac{2a}{16} = \frac{2.10^{-2}}{16} = 0{,}00125m\ .$$

The mesh generated contains 11992 elements and 24864 nodes (Fig.7). The mid-side node is in an ¼ position. The numerical results are shown in Listing2.

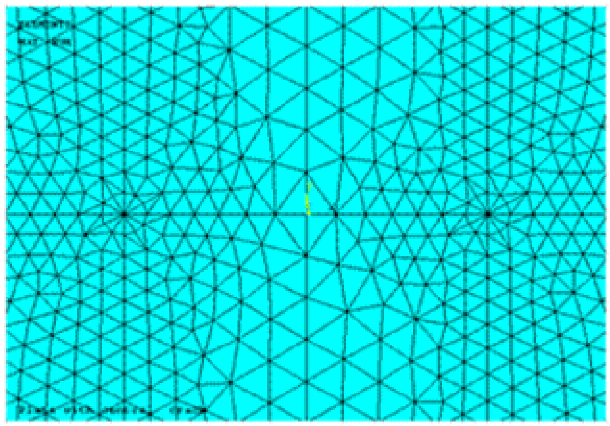

Figure 6 Mesh, r=2a/16

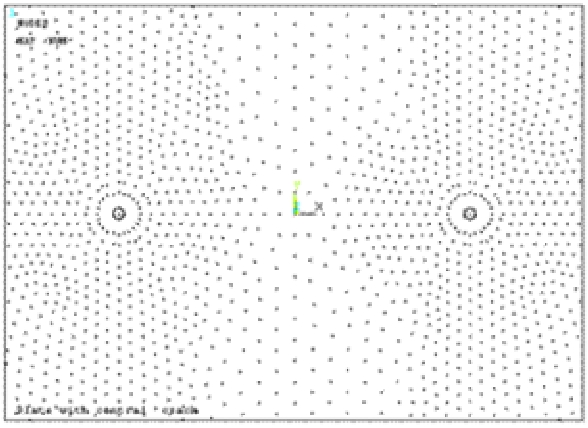

Figure 7 Nodes of the mesh, r = 2a/16

$$\Delta K_{I_2} = \frac{K_I - K_{I\ numerical\ value}}{K_I} \cdot 100 =$$
$$= \frac{563.30 - 561.95}{563.30} \cdot 100 = 0.23\%$$

Mesh №3

The mesh (Fig.8) is generated as follows: the radius of the first row of elements generated in the vicinity of the crack tip is reduced twice additionally:

$$r = \frac{a}{32} = \frac{2.10^{-2}}{32} = 0{,}000625m\ .$$

The mesh generated contains 18454 elements and 37928 nodes (figure 9). The mid-side node is in an ¼ position.

The numerical results are shown in Listing 3.

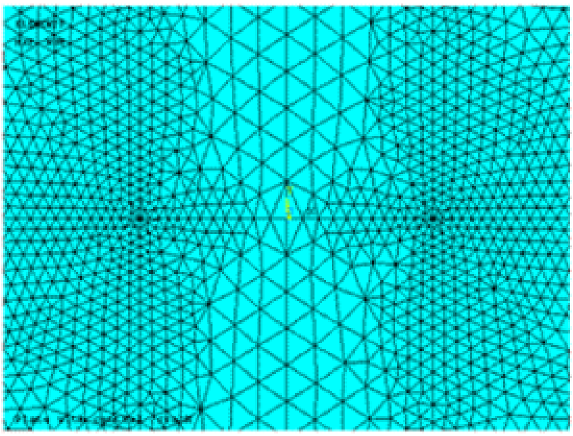

Figure 8 Mesh, r=2a/32

$$\Delta K_{I_3} = \frac{K_I - K_{I\ numerical\ value}}{K_I} \cdot 100 =$$
$$= \frac{563.30 - 561.76}{563.30} \cdot 100 = 0.27\%$$

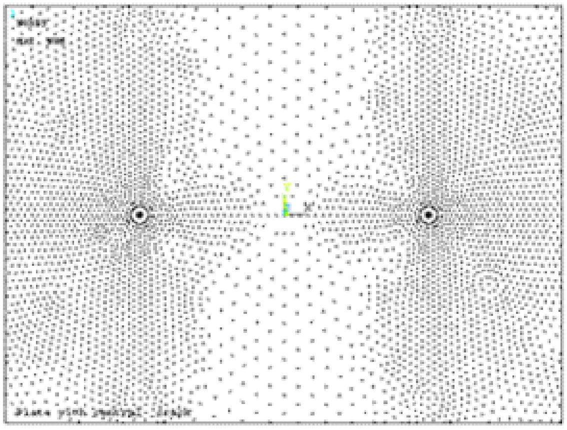

Figure 9 Nodes of the mesh, r = 2a/32

Mesh №4

Elements used are classical ½ point

The mesh (figure 10) is the same as this one presented on figure 8 but the mid side node is in ½ position. Nodes of this mesh are shown on figure 11. Simple isoparametric elements are used.

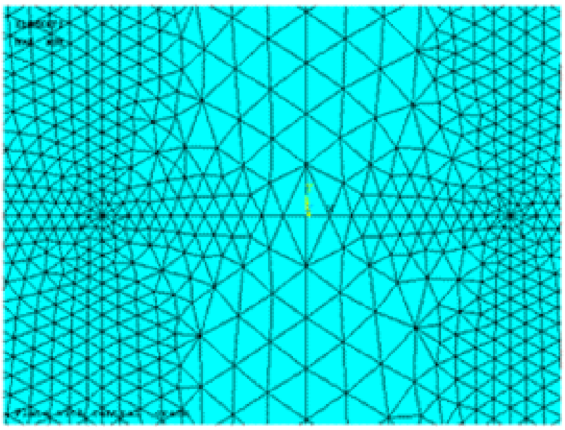

Figure 10 Mesh, r=2a/32

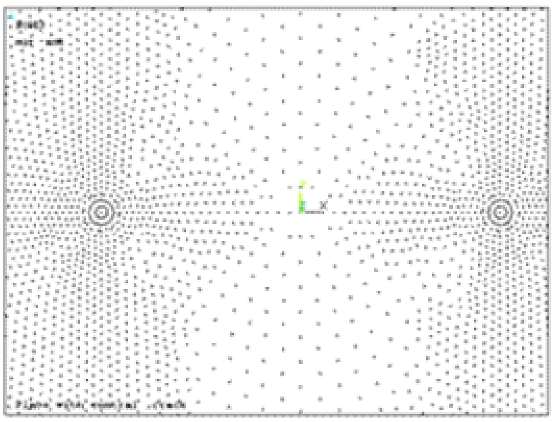

Figure 11 Nodes of the mesh, r=2a/32

The numerical results are shown in Listing4.

$$\Delta K_{I_4} = \frac{K_I - K_{I\ numerical\ value}}{K_I} \cdot 100 =$$

$$= \frac{563.30 - 489.44}{563.30} \cdot 100 = 13.11\%$$

A significant difference between results obtained by the two approaches obviously exists!

4. Conclusions

A numerical simulation for cracked plates with a different mesh size is accomplished by means of ANSYS. The Stress Intensity Factor is received for plates with a centred transverse crack and under uniaxial loading.

The comparison between numerical results and analytical solution shows excellent agreement for KI when the mid-side node is in an ¼ position (when ¼ point elements are used).

The value of the radius of the first row of elements plays the essential role for numerical results for SIF. The ANSYS recommendation for the value of the radius is more accurate (r=2a/8). The simple isoparametric elements are not appropriate for calculating SIF and their usage has to be avoided.

ANSYS gives fictitious results about KII and these results must be neglected. When the plane of the geometrical symmetry coincides with the plane of loading symmetry, then in this plane - tangential stresses are zero and hence KII must be considered to be zero. This is because of the full plate model usage. Such results are not obtained in case of a ¼ geometric model of the plate.

```
CALCULATE MIXED-MODE STRESS INTENSITY  FACTORS
****

   ASSUME PLANE STRESS CONDITIONS

   ASSUME A FULL-CRACK MODEL (USE 5 NODES)

   EXTRAPOLATION  PATH  IS  DEFINED  BY  NODES:
7480    7575    7576   21861   21860
    WITH NODE    7480 AS THE CRACK-TIP NODE

   USE MATERIAL PROPERTIES FOR MATERIAL NUMBER
1
    EX =      0.86000E+09    NUXY =      0.30000
AT TEMP =  0.0000

   PRINT THE LOCAL CRACK-TIP DISPLACEMENTS

   CRACK-TIP DISPLACEMENTS:
        UXC  =-0.34550E-07    UYC=  0.37123E-06
UZC= 0.78886E-30

      NODE   CRACK FACE    RADIUS       UX-UXC
UY-UYC       UZ-UZC
      7480      TIP        0.0000       0.0000
0.0000       0.0000
      7575          TOP             0.62500E-03
0.25641E-08  0.25670E-07   0.0000
      7576          TOP             0.25000E-02
0.93074E-08  0.48601E-07   0.0000
     21861          BOT             0.62500E-03
0.25852E-08 -0.25657E-07   0.0000
     21860        BOT          0.25000E-02   0.
93535E-08 -0.48574E-07   0.0000

   LIMITS  AS  RADIUS  (R)  APPROACHES  0.0  (TOP
FACE) ARE:
        (UX-UXC)/SQRT(R)   =  0.74701E-07  (UY-
UYC)/SQRT(R) = 0.10451E-05
        (UZ-UZC)/SQRT(R) =  0.0000

   LIMITS AS RADIUS (R) APPROACHES 0.0 (BOTTOM
FACE) ARE:
        (UX-UXC)/SQRT(R)   =  0.75519E-07  (UY-
UYC)/SQRT(R) =-0.10445E-05
        (UZ-UZC)/SQRT(R) =  0.0000

 ****   KI =    563.07      ,    KII =   0.26398
,   KIII =   0.0000      ****
```

Listing 1

```
CALCULATE MIXED-MODE STRESS INTENSITY  FACTORS
****

   ASSUME PLANE STRESS CONDITIONS

   ASSUME A FULL-CRACK MODEL (USE 5 NODES)
EXTRAPOLATION   PATH   IS   DEFINED   BY   NODES:
6226    6301    6302   17901   17900
    WITH NODE    6226 AS THE CRACK-TIP NODE

   USE MATERIAL PROPERTIES FOR MATERIAL NUMBER
1
    EX =      0.86000E+09    NUXY =      0.30000
AT TEMP =  0.0000

   PRINT THE LOCAL CRACK-TIP DISPLACEMENTS

   CRACK-TIP DISPLACEMENTS:
UXC  =-0.37767E-07    UYC=  0.37126E-06    UZC=
0.78886E-30

NODE    CRACK  FACE      RADIUS          UX-UXC
UY-UYC       UZ-UZC
      6226      TIP        0.0000       0.0000
0.0000       0.0000
      6301          TOP             0.31250E-03
0.12293E-08  0.18270E-07   0.0000
      6302          TOP             0.12500E-02
0.46212E-08  0.35502E-07   0.0000
     17901          BOT             0.31250E-03
0.12471E-08 -0.18258E-07   0.0000
     17900          BOT             0.12500E-02
0.46598E-08 -0.35522E-07   0.0000

LIMITS AS RADIUS (R) APPROACHES 0.0 (TOP FACE)
ARE:
        (UX-UXC)/SQRT(R)   =  0.49152E-07  (UY-
UYC)/SQRT(R) = 0.10433E-05
        (UZ-UZC)/SQRT(R) =  0.0000
LIMITS  AS  RADIUS  (R)  APPROACHES  0.0  (BOTTOM
FACE) ARE:
        (UX-UXC)/SQRT(R)   =  0.50128E-07  (UY-
UYC)/SQRT(R) =-0.10422E-05
        (UZ-UZC)/SQRT(R) =  0.0000

 ****   KI =    561.95      ,    KII =   0.26298
,   KIII =   0.0000
```

Listing 2

```
 CALCULATE   MIXED-MODE   STRESS   INTENSITY
FACTORS  ****

   ASSUME PLANE STRESS CONDITIONS

   ASSUME A FULL-CRACK MODEL (USE 5 NODES)

   EXTRAPOLATION  PATH  IS  DEFINED  BY  NODES:
9343    9438    9439   27357   27356
    WITH NODE    9343 AS THE CRACK-TIP NODE

   USE MATERIAL PROPERTIES FOR MATERIAL NUMBER
1 EX =      0.86000E+09    NUXY =      0.30000
AT TEMP =  0.0000

   PRINT THE LOCAL CRACK-TIP DISPLACEMENTS

   CRACK-TIP DISPLACEMENTS:
     UXC =-0.35901E-07  UYC= 0.37127E-06  UZC=
0.78886E-30

      NODE   CRACK FACE    RADIUS       UX-UXC
UY-UYC       UZ-UZC
      9343      TIP        0.0000       0.0000
0.0000       0.0000
      9438          TOP              0.15625E-03
0.59870E-09  0.12967E-07   0.0000
      9439          TOP              0.62500E-03
0.23137E-08  0.25521E-07   0.0000
     27357          BOT              0.15625E-03
0.61165E-09 -0.12960E-07   0.0000
     27356          BOT              0.62500E-03
0.23429E-08 -0.25537E-07   0.0000

LIMITS AS RADIUS (R) APPROACHES 0.0 (TOP FACE)
ARE:
        (UX-UXC)/SQRT(R)  =  0.33012E-07  (UY-
UYC)/SQRT(R) = 0.10429E-05
        (UZ-UZC)/SQRT(R) =  0.0000

LIMITS  AS  RADIUS  (R)  APPROACHES  0.0  (BOTTOM
FACE) ARE:
        (UX-UXC)/SQRT(R)  =  0.34004E-07  (UY-
UYC)/SQRT(R) =-0.10419E-05
        (UZ-UZC)/SQRT(R) =  0.0000

 KI =    561.76      ,    KII =   0.0000      ,
KIII =   0.0000
```

Listing 3

```
CALCULATE MIXED-MODE STRESS INTENSITY  FACTORS
****

   ASSUME PLANE STRESS CONDITIONS

   ASSUME A FULL-CRACK MODEL (USE 5 NODES)
   EXTRAPOLATION  PATH  IS  DEFINED  BY  NODES:
9343    9438    9439   27357   27356
    WITH NODE    9343 AS THE CRACK-TIP NODE

   USE MATERIAL PROPERTIES FOR MATERIAL NUMBER
1
    EX =      0.86000E+09    NUXY =     0.30000
AT TEMP =  0.0000

   PRINT THE LOCAL CRACK-TIP DISPLACEMENTS

   CRACK-TIP DISPLACEMENTS:
        UXC  =-0.33799E-07    UYC=  0.37121E-06
UZC= 0.78886E-30

      NODE   CRACK FACE    RADIUS       UX-UXC
UY-UYC       UZ-UZC
      9343      TIP        0.0000       0.0000
0.0000       0.0000
      9438          TOP           0.31250E-03  -
0.17320E-08  0.16719E-07   0.0000
      9439          TOP           0.62500E-03  -
0.16056E-09  0.24584E-07   0.0000
     27357          BOT           0.31250E-03  -
0.17134E-08 -0.16722E-07   0.0000
     27356          BOT           0.62500E-03  -
0.10056E-09 -0.24592E-07   0.0000

   LIMITS  AS  RADIUS  (R)  APPROACHES  0.0  (TOP
FACE) ARE:
        (UX-UXC)/SQRT(R)   =-0.18953E-06   (UY-
UYC)/SQRT(R) = 0.90813E-06
       (UZ-UZC)/SQRT(R) =  0.0000
   LIMITS AS RADIUS (R) APPROACHES 0.0 (BOTTOM
FACE) ARE:
        (UX-UXC)/SQRT(R)   =-0.18982E-06   (UY-
UYC)/SQRT(R) =-0.90824E-06
        (UZ-UZC)/SQRT(R) =  0.0000
 ****  KI =    489.44     ,    KII =   0.77979E-
01,   KIII =   0.0000      ****
```

Listing 4

References

1. Chan, S.K., Tuba, I.S., Wilson, W.K.: *On the Finite Element Method in Linear Fracture Mechanics*. Journal of Engineering Fracture Mechanics, Vol. 2, 1970, p. 1-17, ISSN 0013-7944, Elsevier
2. Byskov, E.: *The Calculation of Stress Intensity Factors Using Finite Element Method with Cracked Elements*. International Journal of Fracture, Vol. 6, June 1970, p. 159-167, ISSN 0376-9429, Springer Netherlands
3. Tracey, D.M.: *Finite Elements for Determination of Crack Tip Elastic Stress Intensity Factors.* International Journal of Numerical Methods Engineering, Vol. 3, April 1971, p. 255-265, ISSN 1097-0207, John Wiley & Sons, USA
4. Tong, P., Pian, T.H., Lasry, S.J.: *A Hybrid Element Approach to Crack Problems in Plane Elasticity.* International Journal of Numerical Methods Engineering, Vol. 7, 1973, p. 297-308, ISSN 0376-9429, Springer Netherlands
5. Henshell, R.D., Shaw, K.G.: *Crack Tip Finite Elements Are Unnecessary*. International Journal of Numerical Methods Engineering, Vol. 9, 1975, p. 459-507, ISSN 0376-9429, Springer Netherlands
6. Barsoum, R.S.: *Triangular Quarter Point Elements as Elastic and Perfectly-Plastic Crack Tip Elements*. International Journal of Numerical Methods Engineering, Vol. 11, 1977, p. 86-98, ISSN 0376-9429, Springer Netherlands
7. Murakami, Y.: *Stress Intensity Factors Handbook-I*. Pergamon Publishing House, Oxford UK, 1990, ISBN 0080348092
8. *** – *Documentation for ANSYS* – Release 10.0