DETERMINATION OF CRACK COEFFICIENTS FOR A SINGLE-EDGE-NOTCH TENSION SPECIMEN USING FRANC2D PROGRAM

BY:

J. R. MOHANTY
NATIONAL INSTITUTE OF TECHNOLOGY, ROURKELA 769008
GUDDY95@GMAIL.COM

ARVIND JONI
NATIONAL INSTITUTE OF TECHNOLOGY, ROURKELA 769008

P. K. RAY
NATIONAL INSTITUTE OF TECHNOLOGY, ROURKELA 769008

B. B. VERMA
NATIONAL INSTITUTE OF TECHNOLOGY, ROURKELA 769008

ABSTRACT

The INSTRON DaDN software has the provision to use the compliance method of crack growth measurement for standard specimens. The relationship between compliance and crack length is given in the form of a polynomial equation whose coefficients assume different values for different specimen geometries. In the present work an attempt has been made to determine the coefficients for a non-standard (single-edge-notch tension) specimen using experimental data and also by Finite Element Method (FRANC2D software). It is found that the coefficients obtained by Finite Element method gives better crack length estimates for the above specimen geometry in comparison to the coefficients obtained experimentally.

INTRODUCTION:

The "INSTRON" fast track da/dN fatigue crack propagation program is used for online crack growth monitoring and analysis for specimens with standard geometries like Compact Tension (CT) specimen, Middle-cracked Tension (MT), Single Edge Notch Bend (SENB), Disk Compact Tension (DCT). The da/dN program has the provision to determine crack growth by various methods such as visual, A.C. potential drop, D.C. potential drop and compliance. In the present work an attempt has been made to use the compliance method of crack growth measurement. The relationship between compliance and crack length is given in the form of a polynomial equation. The relationship is usually expressed in terms of the dimensionless quantities of compliance, EVB_{eff} / p and the normalized crack length a/w. The relationship is given by

$$\frac{a}{w} = \sum_{i=0}^{i=5} C_i * U_x^i$$

where C_i's are the compliance coefficients. The values of these coefficients are incorporated in the software for standard specimens mentioned above. In the present case, the specimen is a non-standard one of the type Single Edge Notch Tension (SENT)[2]. Hence it is necessary to determine the coefficients and incorporate them in the software for online determination of crack growth. Here coefficients have been determined experimentally by performing tests and also analytically by using Finite element package "FRANC2D" and finally the results are compared. The FRANC2D is a finite element based simulator for curvilinear crack propagation in planar (plane stress, plane strain, and axisymmetric) structures. CASCA which is used for the creating the meshes, is a pre-processor for generating initial input files for FRANC2D.

SPECIMEN DETAILS AND GEOMETRY:

Single edge notched Tension (SENT) specimens of dimension 52 mm x 170mm x 6.5 mm were prepared as per details given in Fig.1. The three holes at the top and three holes at the bottom were provided to facilitate loading in mode I / mixed mode / mode II directions in subsequent tests.

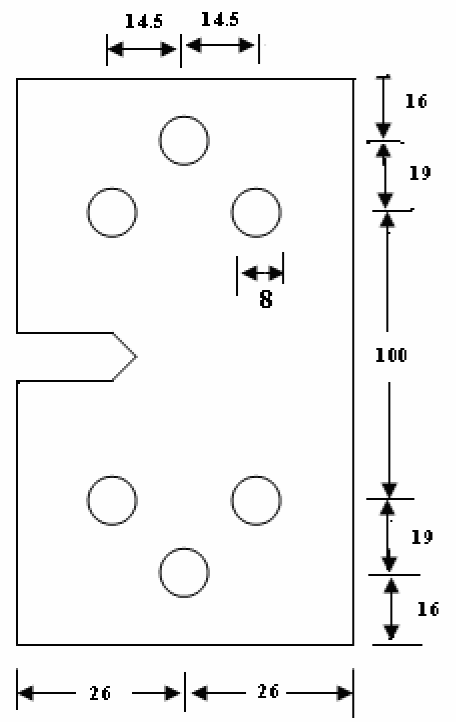

Fig.1 Dimension of the Model

The material used in the investigation is Steel having composition shown in Table.1 below.

Table 1.Composition(%) of the material tested:

C	Mn	P	S	Si	Al	Nb	V
0.16	1.40	0.023	0.006	0.33	0.034	0.039	0.036

The elastic modulus of the specimen material is 2 x 10^5 MPa and Poisson ratio is 0.33.

EXPERIMENTAL PROCEDURE:

The fatigue tests were performed in the L.T direction. Before fatigue test, the notched specimens were precracked upto a length of 16mm (including a notch length of 13 mm). Precracking was done according to ASTM-E647-88a standard. After precracking the specimens were cycled in an INSTRON-8502 in tension-tension constant stress amplitude mode at stress ratio R=0.1. The constant amplitude loading tests were done under a sinusoidal waveform loading.

CALCULATION OF COEFFICIENTS TO PREDICT THE CRACK LENGTH IN DADN SOFTWARE:

The Crack Opening Displacement (COD) gauge which can be fixed to the notch using knife edge can be generally used to determine the length of growing crack by means of coefficients used in DaDN software. It determines the length of growing crack based on the changes in the crack opening level with maximum stress level. The coefficients usually vary according to the geometry of the specimen. Since the geometry of the specimen which we are using is not a standard one, it is necessary to determine the value of coefficient to standardize the specimen.

The general formula used in software for calculation of crack length:

$$a/w = C_0 + C_1 U + C_2 U^2 + C_3 U^3 + C_4 U^4 + C_5 U^5$$

Where C_0, C_1, C_2 …….. are the coefficients to be calculated,

and $U = (1+ \sqrt{(Evb/p)})^{-1}$

Where E= Modulus of elasticity

v=Linear load displacement

b=Thickness of specimen

p= Applied load.

From main cracking experiment, the reading of the number of cycles corresponding to the marked crack length was noted, and then the value of Evb/p corresponding to the number of cycles noted down from tabulated data of computer. This was done for at least six markings of crack length to find different values of U_i.

The coefficients can be calculated by using MATLAB or NM-Tool kit or any other numerical analysis software.

SIMULATION PROCEDURE:

Assumption:

To simulate the behavior of crack growth on the specimen some assumption and approximations are required. Here analysis was undertaken based on the assumption that the tensile load is acting uniformly on the specimen perpendicular to the crack propagation.

MODELING PROCEDURE:

A mesh generating program CASCA, which is distributed with FRANC2D, was used to create the initial mesh configuration for FRANC2D simulations. Other mesh generating programs can also be used, provided a translator is available to convert the mesh description to the FRANC2D *.inp format. The procedure for creating a mesh is straight forward, as illustrated in figure. To begin with the problem, outline is first created followed by the division of the sub regions within the problem boundary as shown in fig.2.. Prior to assigning the type of elements to each of the sub regions, the boundaries for all the sub regions are divided into the required number of segments as shown in fig.3.

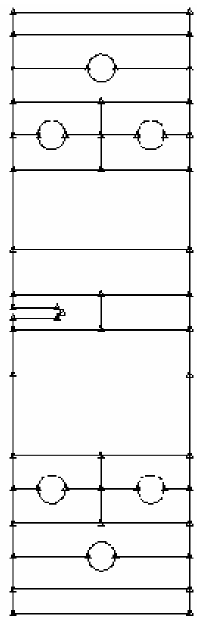

Fig.2.Sub-region within the problem boundary.

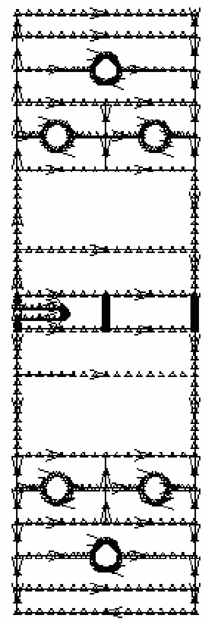

Fig.3. Sub-regions divided into required number of segments

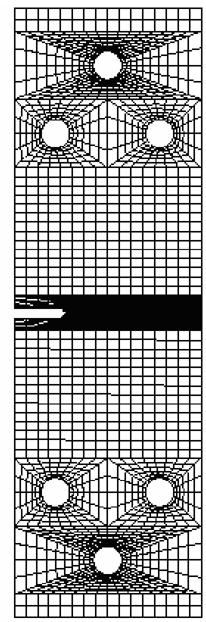

Fig.3a. Resulting mesh of the present simulation

Figure.3a. shows one of the mesh used in the present study which comprises bi-linear four sided elements and the boundary conditions are specified by fixing lower half of all the three holes by FIX EDGE XY option. Fig.4. shows the resulting mesh for the present simulation after applying the boundary conditions.

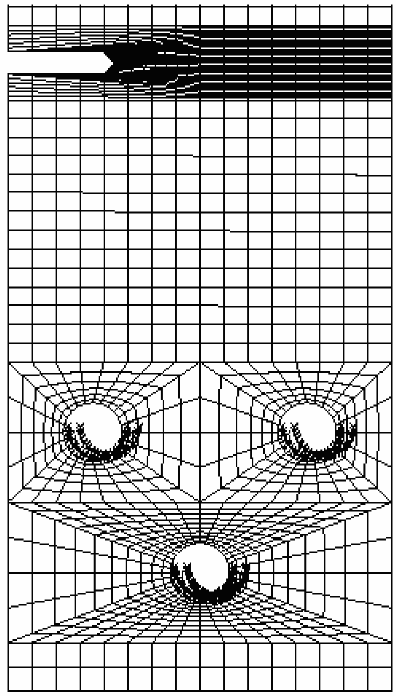

Fig.4. Fixity of lower half of the holes.

DESCRIPTION OF LOADS:

In a real case, loading is due to pin. Since it is a complicated process, for the present analysis, model the pin as a distributed load over a portion of the surface of hole. Here the applied load is distributed normally because pressure due to pin acting normal to the surface of pin hole. Upon different options of load distribution, pin loading can be best modeled by a quadratic load distribution over one half of pin hole which is as shown in fig.5. Uniform distribution of total load on the three holes is taken as 16.63 KN.

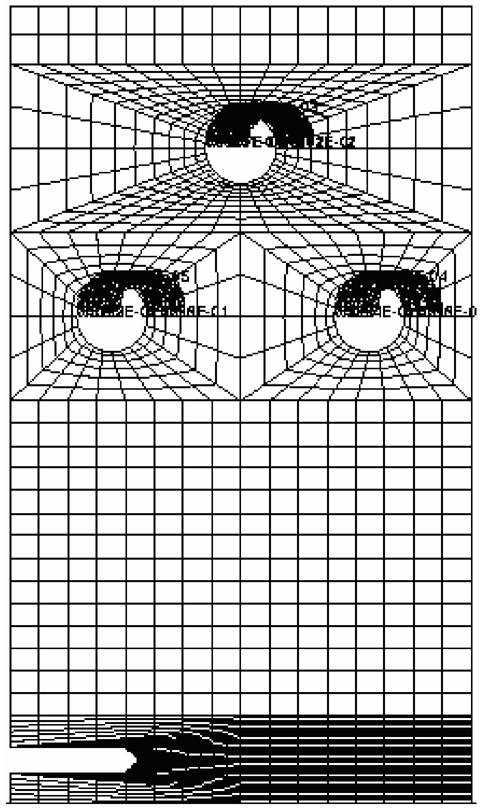

Fig.5. Load distribution in the three holes.

MATERIAL PROPERTIES:

The properties of model used in the present study are shown in table.2:

Table.2. Properties of the model.

Material type	Isotropic
Youngs Modulus	$2*10^5$ MPa
Poisson ratio	0.33
Thickness	6.5 mm

CRACK PROPAGATION:

In order to simulate the crack growth, an initial non cohesive crack was placed at the notch tip, where it was predicted that the critical tensile stress would occur. Having specified the location of crack, predict the direction in which the crack would propagate. This direction is the direction of maximum hoop stress around the crack tip. Prior to performing the analysis, it is necessary to specify the crack tip. In the present analysis using FRANC2D, the crack tip coordinate is specified corresponding to the lines marked on the specimen surface during experimental study. Then the crack propagation can be performed using standard method. Fig.6. shows the crack propagation after application of load.

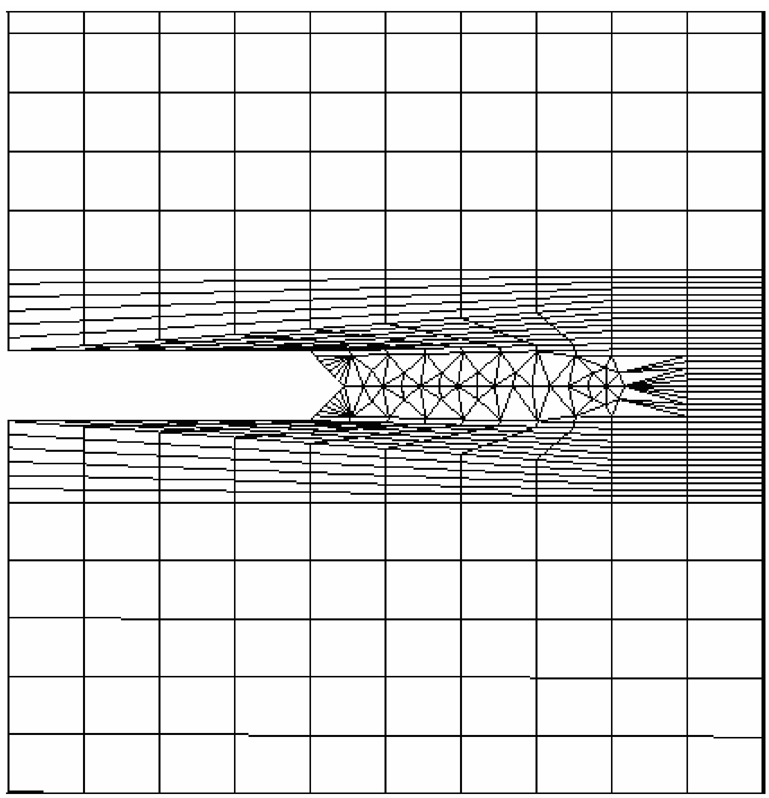

Fig.6. Crack propagation due to the application of load

RESULTS AND DISCUSSIONS:

The tabular data of displacements(v) and coefficients (C_i) corresponding to different crack lengths and also the experimental data are given in table.3 for comparison of results.

Table.3. Displacements and coefficients corresponding to crack length

Crack Length (a)	Displacement (v) data from machine	Calculated Coefficients from column-II	Displacement (v) data of COD gauge position from FRANC2D	Calculated Coefficients from column-IV
19.23	0.0492	0.7376	0.0586	1.0066
20.28	0.0562	0.32499	0.0660	-2.5363
21.43	0.0652	-7.9138	0.0754	1.5075
22.41	0.0743	4.9771	0.0848	2.8156
23.37	0.0837	42.4170	0.0953	-5.9411
24.24	0.0951	-71.1828	0.1060	-1.5556

After obtaining the coefficients from the finite element simulation and also from visual method using MATLAB, we verified by performing the fatigue test for different SENT specimens. It was observed that the data obtained in later case showed 2mm variation in crack length than the former, which gave exact value.

It has been concluded that the coefficients obtained by FEM simulation using FRANC2D is much better than that of experimental one.

REFERENCES:

1. ASTM E647-88a, Standard test method for measurement of fatigue crack growth rates, 1990.
2. Cornell Fracture Group (1995), FRANC2D; A two dimensional crack propagation simulator – *tutorial and User's guide, Version 2.7.*