Simulation of Photopolymer CT Specimen for Stress Intensity Factors

M Afzal Bhat¹, A A Shaikh²

¹Research Scholar, ²Associate Professor, Department of Mechanical Engineering, SVNIT, Surat, India 395007

Abstract—Now a days product development using prototype methods for actual usage is a growing concept in engineering. Present work deals with simulation of Compact Tension specimen to get fracture parameters J – integral and Stress intensity factors of product built by polyjet technology 3D printer additive manufacturing system. FEA simulation is carried out with the help of Abaqus software by simulating actual testing procedure by applying a tensile load on one of the pins to have mode I fracture. Fracture parameters J – integral and stress intensity factors are requested as history output. A comparison is made between the values of stress intensity factor obtained by simulation and by calculation using ASTM standard. It is observed that the values of stress intensity factor obtained by simulating the specimen in FEA software Abaqus/CAE are near to the values obtained by calculations.

Keywords—stress intensity factor, polyjet technology, photopolymer, 3D printer, additive manufacturing.

I. INTRODUCTION

With the advances in technology and the materials, it has become possible to have functional end products directly from the rapid prototyping systems. Strength of such products have to be investigated in terms of tensile and fracture aspects to realise its reliability for usage and to meet in-service loading and operation requirements. Several layer fabrication techniques have been developed which have been categorized into liquid-based, powder-based and solid-based processes depending on the state of the material being processed during layer fabrication. Examples are stereo lithography, selective laser sintering, fused deposition modeling, etc, while Polyjet is one of the recent additive manufacturing processes.

Polyjet technology uses a jetting head made of micro nozzles to spray liquid material and accurately build each layer at about 16 micron thickness upon solidification by exposure to an ultraviolet (UV) light. The process builds the part layer by layer using two different photopolymer resins as build and support material followed by a UV light bulb to solidify the layers in an additive manner. The first commercialization of such a technique was introduced with Polyjet technology of Objet Geometries in 2001, Gibson (2009) [1].

The fracture properties of the products are important and are studied by various researchers. Shahani et al [2] investigated experimentally the effect of thickness on ductile fracture toughness of plates made of steel alloy GOST 08Ch22N6T. The effect of thickness on the fracture toughness of metallic foil (Double-edge cracked specimens) with thicknesses ranging from 0.02 to 1 mm was investigated experimentally by Kang et al [3]. The effect of specimen notch root radius and specimen size (thickness and ligament) on the apparent fracture toughness is investigated by Mourad et al [4]. Unstandardized experiments of fracture toughness in which single edge notch bend (SENB) specimens of five different thicknesses (4 to 22 mm) are carried out by Wang et al [5] to investigate the size effect on the ductile and brittle fracture. Ono et al [6] fabricated four different sizes of compact tension (CT) specimens of a reduced activation ferritic steel, and measured the fracture toughness at room temperature by means of the unloading compliance method referring to in the ASTM E1820-99a.

In technology brief [7] a standardized compact tension specimen is modeled, and J-integral results are compared with those generated from applicable American Society for Testing and Materials (ASTM) standards and from a laboratory testing method. It is shown that Abaqus results are in very close conformance with the experimental results. Crack-growth simulations of the Arcan test is carried out by Deng and Newman Jr [8] using the finite element code ZIP2DL under plane stress and both small- and large deformation kinematics conditions. Simulation of turbine blade of a conventional power generating plant is done by Morris and Ing [9]. The behavior of Carbon-Epoxy laminated composite is studied numerically by modeling of Arcan specimen at various loading angles and crack lengths in ABAQUS finite element software by Nikbakht et al. [10]. The numerical analysis is performed under a constant load of 1000 N.

Es-said et al [11] studied the tensile strength, modulus of rupture, and impact resistance for different layer deposition angles of ABS prototype solid models and concluded that the samples where deposition is along the length exhibit superior strength and impact resistance.
Recently Vega et al [12] focused on the study of mechanical strength by tensile testing and three point bending and Wiebull analysis is done on the izod impact testing. They concluded that layer orientation and surface roughness has significant effect on penetration of the coating material and overall strength of the samples. More recently Ibrahim and Irfan [13] studied dynamic crack propagation under impact loading by modified Hopkinson Bar of ABS material used in prototyping and they observed that layered ABS material used in prototyping offer better resistance to crack propagation as compared to monolithic ABS.

In the current study simulation of compact tension specimen is carried out for photopolymer material specimen in FEA software Abaqus to obtain fracture property - $J$ - integral. From these $J$ - integral values stress intensity factor $k$ is calculated and compared with the values of $k$ obtained by calculating using equations as discussed in section 4.

II. Finit Element Analysis Simulation

The dimensions of the specimen under investigation is shown in figure 1. The initial crack length is 25 mm and the thickness of the specimen is 8mm. The elastic modulus of the specimen is 2500 MPa and Poisson’s ratio is 0.3.

![Figure 1: Dimensions in mm of Compact Tension Specimen used for fracture testing.](image)

A two-dimensional plane-strain model is analysed in Abaqus. The loading pins are modelled as rigid bodies. The specimen is loaded by applying a concentrated load to the upper pin in the vertical direction, all other motions of the pin are restrained. The lower pin is constrained as fixed to restrict all motions. Two analysis steps are used. In the first step the lower pin is constrained.

In the second step controlled load is applied gradually on the upper pin. The crack is introduced into the model with a seam. A seam is defined in the model as an edge in a two dimensional part that can open during an analysis. From the crack editor, the virtual crack extension direction is specified with the $q$-vector. The partitioned geometry is shown in figure 2 also seam is represented as bold black line and $q$-vector for current analysis is taken in the direction of arrow.

![Figure 2: Seam and q-vector direction.](image)

For a sharp crack the strain field becomes singular at the crack tip. Including the singularity at the crack tip for a small- strain analysis improves the accuracy of the $J$ - integral, stress intensity factors, and the stress and strain calculations. The geometry is partitioned by defining the circular lines centred on the crack tip for facilitating the generation of a focused mesh. The crack tip is meshed using a ring of collapsed quadratic quadrilateral elements. Second-order elements are generally used to obtain a mesh singularity at the crack tip. In the present analysis a value of 0.25 is used for the midside node parameter which moves the midside nodes on the element sides adjoining the collapsed edge to the 1/4 points of the elements.

At the crack tip the element sides are collapsed with single-node- type degenerate element control. The circular partitioned areas are meshed using the swept meshing technique. This method allows the mesh to be regular and focused. The remaining portion of the model is free meshed using the medial axis meshing algorithm. The edge-based tools for specifying mesh seeding facilitate the development of a focused mesh around the crack tip. The mesh used in the present analysis is shown in figure 4. In the second analysis step contour integral output is requested for 10 contours.
III. RESULT AND DISCUSSIONS

A. Stress Intensity Factor

The specimen with compact tension geometry held in clevis is loaded gradually in tension. The loading of the specimen is done in mode I manner where the crack surfaces move directly apart. Load versus load line displacement is recorded and Peak load of each specimen is noted from these Load vs load-line displacement curves for each specimen. With the increase of thickness, an increase in the peak load is observed.

The peak load is used to calculate stress intensity factor \( K_Q \) using the equations 1 and 2 as per ASTM standard D5045 - 99.

\[
K_Q = \left( \frac{P_Q}{BW^{1/2}} \right) f(x) \\
f(x) = \frac{(2+x)(0.886+4.64x-13.32x^2+14.72x^3-5.6x^4)}{(1-x)^{3/2}}
\]

For a specimen with the value of width \( W \) of 50 mm and the value of \( x = 0.5 \), where \( x \) is the ratio of crack length \( a \) and the width \( W \) of the specimen while the thicknesses \( B \) varies from 4 mm to 20 mm with an increment of 4mm, Stress intensity factor is calculated and tabulated in Table I.

<table>
<thead>
<tr>
<th>S.No</th>
<th>Thickness (mm)</th>
<th>Load (N)</th>
<th>Stress Intensity factor (MPa.mm(^{1/2}))</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>04</td>
<td>54.5</td>
<td>18.6118</td>
</tr>
<tr>
<td>2</td>
<td>08</td>
<td>301.3</td>
<td>51.4470</td>
</tr>
<tr>
<td>3</td>
<td>12</td>
<td>534.3</td>
<td>60.8211</td>
</tr>
<tr>
<td>4</td>
<td>16</td>
<td>784.0</td>
<td>66.9340</td>
</tr>
<tr>
<td>5</td>
<td>20</td>
<td>927.0</td>
<td>63.3141</td>
</tr>
</tbody>
</table>

Stress intensity factor \( K_Q \) verses specimen thickness are plotted in figures 5. The value of \( K_Q \) first increases with the increase in thickness, but with further increase in thickness the value of \( K_Q \) decreases. The maximum value of stress intensity factor is observed for specimen with thickness of 16 mm.

Figure 5: Thickness vs stress intensity factor plot for average values.

B. Simulation Results

The crack is defined in the Interaction module of Abaqus CAE software, the crack front and the crack tip is also specified. Contour integral output is requested for 10 contours and the value of J-integral is taken for 5th contour.
All the elements within the crack front and one layer of elements outside the crack front are used to determine the first contour integral. Abaqus automatically adds a single layer of elements to the group of elements that are used to calculate the previous contour integral for computing the additional contour integrals. The results obtained from Abaqus for the J-integral and Stress intensity factor are tabulated in table II.

### TABLE II

<table>
<thead>
<tr>
<th>S.No</th>
<th>Thickness (mm)</th>
<th>Load (N)</th>
<th>( J )-integral (N.mm)</th>
<th>Stress Intensity factor (MPa.mm(^{1/2}))</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>04</td>
<td>54.5</td>
<td>13.2</td>
<td>17.462</td>
</tr>
<tr>
<td>2</td>
<td>08</td>
<td>301.3</td>
<td>106.4</td>
<td>49.5766</td>
</tr>
<tr>
<td>3</td>
<td>12</td>
<td>534.3</td>
<td>149.1</td>
<td>58.6874</td>
</tr>
<tr>
<td>4</td>
<td>16</td>
<td>784.0</td>
<td>180.5</td>
<td>64.5721</td>
</tr>
<tr>
<td>5</td>
<td>20</td>
<td>927.0</td>
<td>161.5</td>
<td>61.0790</td>
</tr>
</tbody>
</table>

The values of stress intensity factor is calculated from

\[ J = \frac{k^2}{E} \]

Where E is young’s modulus of the material used for specimen. The lower pin of the specimen is constrained to restrain all moments while the upper pin is allowed to move in load applied direction. The distribution of the stresses around crack tip is shown in figure 6. It is observed that the stress is distributed symmetrically around crack plane.

Figure 6: Distribution of stresses around crack tip.

IV. CONCLUSIONS

Rapid Prototyping being used as actual product, hence fracture study is focussed primary in present work. Compact tension specimens are simulated in FEA software Abaqus to investigate fracture properties of material used in rapid prototyping system 3d printer. The following observations are made:

- The mode I stress intensity factor increases with increase in thickness from 4 mm to 16 mm, then it decreases with further increase in thickness to 20 mm.
- The peak value of mode I stress intensity factor is attained when the specimen thickness is 16 mm with the value of 68.2146 MPa.mm\(^{1/2}\).
- Stress intensity factor calculated from the FEM simulation results are found close to the values of stress intensity factor obtained by calculations from ASTM standard.

REFERENCES


