

# Material Analysis and Abaqus Material Card Generation of an Fiber Reinforced PP Plastic Material

Naveen Kumar Rawat  
Student of M.Tech. (Mech.)  
B.V.C.O.E  
Pune, India

Prof. V. K. Kurkute  
Mechanical Department  
B.V.C.O.E  
Pune, India

Dipl.- Ing. Christian Bresser  
Mechanical Simulation  
Huf Hülbeck & Fürst GmbH & Co.KG  
Velbert, Germany

**Abstract**— This work presents the creation of Abaqus material cards for fiber reinforced PP plastic. A comparison between different material cards such as Elastic, Elastic Plastic, Elastic Plastic with Fiber orientation and Elastic-plastic with damage were prepared and compared by using software such as Abaqus 6.14-2 and Hyperworks v 13.0.

The material testing of PPGF30 were done according to ISO 527 and the material properties in the form of engineering stress and engineering strain were obtained for parallel and perpendicular to the fiber orientation. Now true stress, true strain and effective plastic strain are found for parallel as well as in perpendicular direction. Different material cards are prepared and simulation setup in Hypermesh v.13.0 is done. After that an Abaqus run for each prepared material card is made and their behavior is analyzed in terms of von-mises stress, contour plots of Von-mises stress and their respective stress triaxiality.

**Keywords**—Composite material, material testing, material cards, stress triaxiality, von-mises stress, DUCTCRT

## I. INTRODUCTION

A composite material is a structural material that consists of two or more combined constituent which are combined at macroscopic level and are not soluble in each other. It consists of fiber and matrix phase. Here we would be analyzing the behavior of short fibers reinforced PP material. Our scope of study is of glass fibers, chemically known as silicon dioxide. As the name implies, glass has been spun into the form of fibers. Fiberglass in non-conductive, hence good choice for electrical appliances. For matrix we would be using polypropylene. Polypropylene is an inexpensive material and it offers many advantages. The biggest advantage is that it is light in weight and possesses excellent resistance with a low moisture absorption rate.

## II. MECHANICAL BEHAVIOUR OF SHORT FIBERS

Materials can be isotropic, anisotropic, and homogenous or heterogenous. Most general engineering materials such as metals, plastics or ceramics are typically considered to be isotropic. Whereas most composite materials are neither homogeneous nor isotropic. Composites are inhomogeneous (or heterogeneous) as well as non isotropic in nature. When defining isotropic material, we need to define 2 independent elastic constants (young modulus of elasticity and poisson's

ratio. Similarly for orthotropic material 9 independent constants and for transversely isotropic material 5 independent constants need to be defined.

## III. FIBER ORIENTATION AND FRACTURE MECHANICS

It has been observed that fiber orientation has a substantial effect on the properties of composite material. The short fibers which are randomly oriented behave as isotropic in nature. But as the orientation becomes more aligned the properties become more anisotropic. It has been observed that the properties are much stronger in direction of fiber orientation than to the transverse direction. The graph in Figure 1 demonstrates this and it can be seen that the elastic modulus is much higher in the direction of fibre orientation than in the transverse direction.

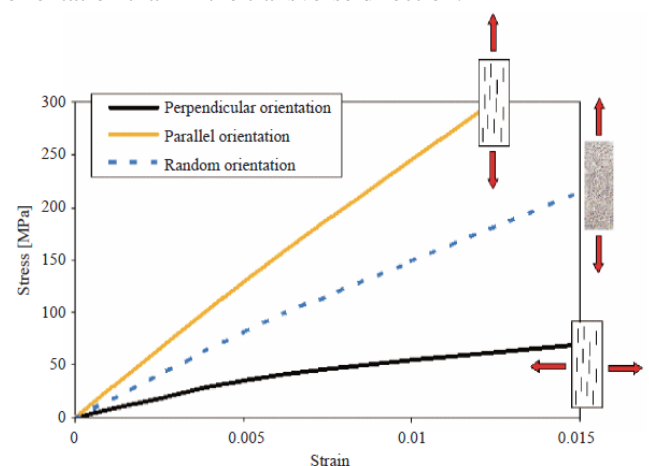


Fig. 1 Stress-strain graph

In the uniaxial tensile test the neck produces a triaxial stress state in the center of the specimen, which promotes void nucleation and growth in the larger particles. Upon further strain, the voids coalesce, resulting in a penny-shaped flaw. The outer ring of the specimen contains relatively few voids, because the hydrostatic stress is lower than in the center. The average stress (Hydrostatic stress) is more in necked section than the other sections. After the necking in uniaxial tension test, the stress is no longer constant in the necked section of the specimen. The deformation during this phase gets localized in a neck

region of the specimen. The assumption of constant axial strain is not valid as it changes from section to section, so to define the state of material by using von-mises stress and equivalent plastic strain would not be proper. After necking, we have to define values of stress triaxiality and fracture strain. It is assumed that the equivalent plastic strain at onset of damage is a function of stress triaxiality and strain rate.

#### IV. ABAQUS MATERIAL MODEL

In Abaqus to model the elastic plastic behavior, we have to select a suitable plasticity model from one of the plasticity behaviors categories and an elasticity material model from one of elasticity categories. There are different materials models available for elastic as well as plastic behavior in Abaqus. For our case we would be using linear elasticity model and classical metal plasticity model. Now to consider the effect of damage, progressive damage and failure model is used. For this criterion, we need to define effective material response, damage initiation criteria and damage evolution law.

#### V. MATERIAL TESTING

For determination of tensile properties of plastics and fiber reinforced plastics "ISO 527" is being used. Our focus of interest is ISO 527-1 and ISO 527-4. In tensile testing, the specimen needs to be elongated in its major longitudinal axis at a steady speed until the test specimen breaks or until the stress (load) or the strain (elongation) reaches some predetermined value. During this procedure, the load sustained by the specimen and the elongation are measured. These tests were done at the speed of 1mm/sec at the room temperature of 23 degree Celsius. The geometry of the specimen with a thickness of 2 mm is shown in the figure 2:

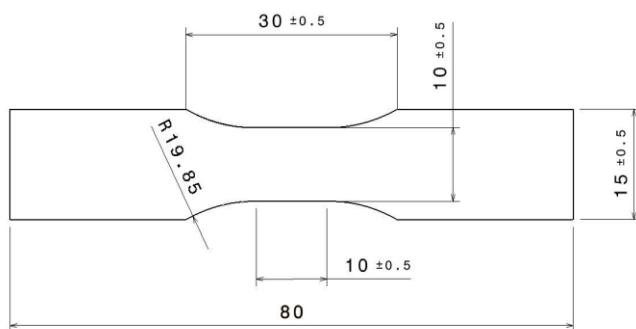


Fig. 2 Geometry of specimen tested

From the test results, we get the engineering stress and engineering strain for the specimen as shown in figure 3. The results are for along the load direction (parallel) and perpendicular to the load direction. As can be seen the engineering stress in along the direction is comparatively high as compared to the perpendicular direction

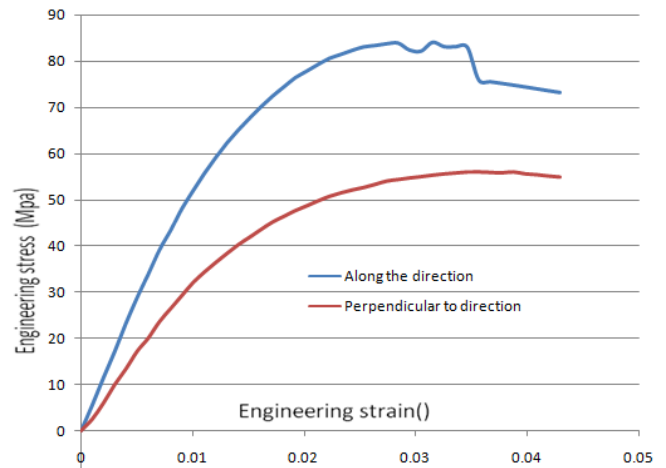


Fig. 3 Engg. Stress versus Engg. Strain for parallel and perpendicular direction

Now the next step is to calculate the modulus of elasticity in both the directions. We have used chord slope method for it according to ISO 527. Young modulus for parallel direction comes out to be 5766 Mpa and 3363 Mpa in perpendicular direction. As can be seen the young modulus is more in parallel direction as compared to perpendicular. The next step is inputting to Abaqus, we need to convert these engineering values to elastic plastic material card.

#### VI. MATERIAL CARD CREATION

We have obtained engineering stress and strain values from material testing. Now our focus is to convert these material properties into suitable form, so as to input the values into Abaqus. The first step is to find the true stress and true strain.

When we are solving a linear static analysis; engineering values are sufficient but for non-linear analysis, we have to input true stress-strain values. Hence Calculations of true stress and true strain are done for both the parallel and perpendicular direction using the below formulae:

$$\text{True stress} = \text{eng stress} (1 + \text{eng strain}) \quad (1)$$

$$\text{True strain} = \ln (1 + \text{eng strain}) \quad (2)$$

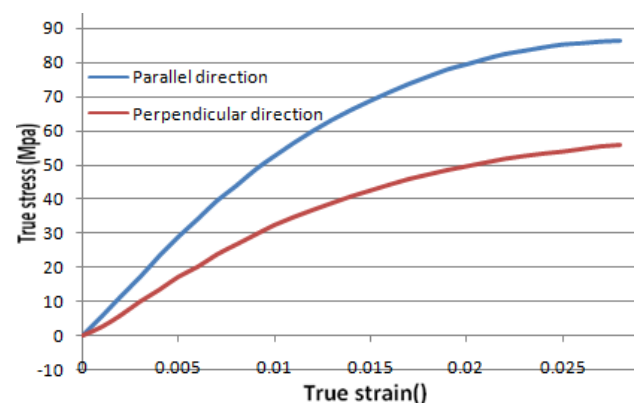


Fig. 4 True stress versus true strain for parallel and perpendicular direction

As already discussed, we need to input elastic and plastic properties separately in Abaqus. So to do that we need to define true stress and logarithmic plastic strain. We have already found true stress and true strain for parallel and perpendicular direction. The effective plastic strain  $\epsilon_{in}^{pl}$  is found by using equation 3:

$$\epsilon_{in}^{pl} = \ln(1 + \epsilon_0) - \frac{\sigma_{true}}{E} = \text{True strain} \quad (3)$$

Now we need to define the yield point for the material. To define the behavior after the yield point, we need to use the classical metal plasticity. It is important to note that plastic strain values are used instead of total strain values. Also, the first pair of data should correspond to the beginning of plasticity (the plastic strain value must be zero in the first pair). The first set of data defines the initial yield stress of the material; hence first value should be zero. So first we need to find or define the yield point upto which the stress is proportional to strain and it has a constant young modulus of elasticity.

The figure 5 represents the curve plotted for true stress and true strain values. A red line is drawn such that it is able to capture maximum number of points from the curve. The final point captured represents the yield stress of the material. This final point has the stress value= 28.9365 Mpa and corresponding true strain of 0.005

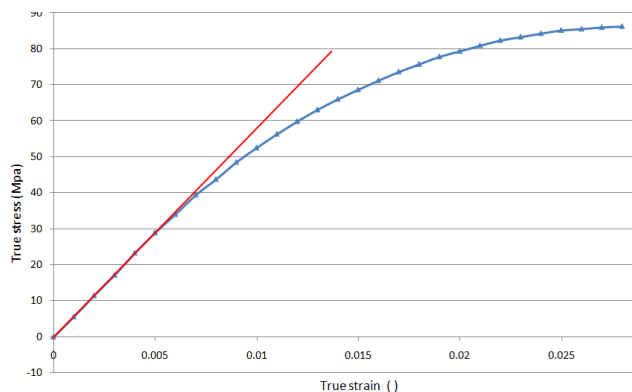


Fig. 5 Calculation of yield point for along the direction

We have got a plastic strain value of -0.00002 for the stress value of 28.9365 Mpa. This is the first point for the plastic behavior. As discussed, the first set of data point for plastic model should be zero. Hence first data point for true stress versus equivalent plastic strain is (28.9365 Mpa, 0.000). The averaged material card is found by averaging the values in parallel and perpendicular direction as shown in the figure 6

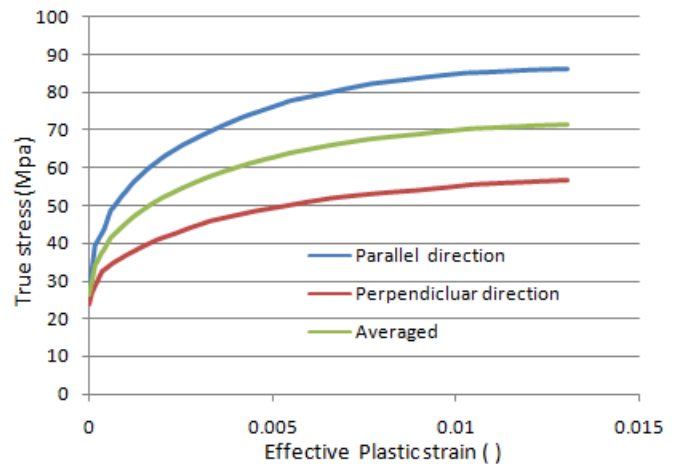


Fig. 6 Comparison of Averaged, parallel and perpendicular material

The elastic plastic material card tells us about the state of material in terms of true stress and effective or equivalent plastic strain. Now to consider fracture mechanics i.e. damage, we need to define stress triaxiality and fracture strain. That is state of material is known by its stress triaxiality and fracture strain. Hence to define damage, we need to define fracture locus of stress triaxiality and fracture strain. We know the stress triax values and corresponding to them, we need to find the fracture strain.

The fracture strain is plastic strain, given by = total strain of material – elastic strain of material. These fracture strain values are obtained from experimental data. Consider the state of tension, here stress triax=0.33 and its corresponding fracture strain=0.03 obtained from tensile experimental data. For compression, it is assumed that no failure occurs. Hence a large value of fracture strain is guessed for compression states. Similarly other values are calculated or guessed and a fracture locus is defined as shown in the fig. 7

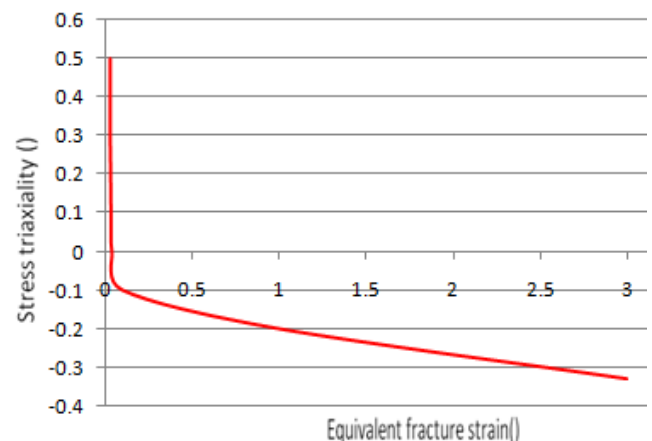


Fig. 7 Fracture locus of stress triaxiality versus equivalent fracture strain

Now different material cards were created namely “Elastic only”, “elastic plastic”, “Elastic plastic with fiber orientation” and “Elastic plastic with damage”.

## VII. ABAQUS SIMULATION SETUP

The Abaqus finite element system includes Abaqus/Standard and Abaqus/Explicit algorithm. We are using Abaqus/Explicit as it is better suited to damage model. Abaqus simulation setup generally consists of preprocessing phase, solution and post processing phase. For preprocessing, we are using Hypermesh v13.0. For processing or solution part, we have used Abaqus 6.14-2. Output data in the format ".inp" is provided to the Abaqus command and a run is made. Post-Processing means viewing the results. It may be in form of contour plots of stress, strain, displacement, etc. Also graphs can be plotted of different variables as required. For post processing part, we have used Hyperview v 13.0. The ".odb" file generated from Abaqus is imported and results are viewed and interpreted.

The different steps in the preprocessing phase are geometry modeling of tensile specimen, inputting material and property information, defining boundary or loading conditions and giving output requests. The below figure 8 shows the geometry of the specimen made in Hypermesh.

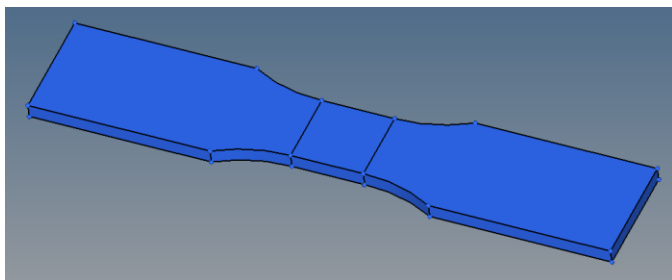


Fig. 8 Geometry modeling in Hypermesh

Now the next step is to mesh the model. We have used hexahedral mesh of element type C3D8R. The figure 9 represents the meshed model of the tensile specimen.



Fig.9 Hexahedral mesh of the specimen

The next step is to input material and property information. Before that we need to set the user profile as Abaqus/Explicit.

We will be simulating the tensile testing of specimen with the created material cards. The material card is the only part which will be needed to change for different runs. All other defined parameters would be kept same. Now we will be defining property for the meshed part. The property module contains information about the section that is whether it is shell, solid, rigid etc. Here we have to define "solid section" as we are using 3D elements. Also material is always assigned to property first and then property is assigned to the meshed component.

Now boundary conditions were applied. We will be fixing one end and at the other end a "displacement of 3.0 mm" will be applied. To apply the displacement gradually for the taken time period we need to define a curve. Let us take the total time period for simulation as "0.3 seconds". Now we will create a curve namely "amp" and its coordinates are defined as shown below:

```
*AMPLITUDE, NAME =amp, DEFINITION = TABULAR
0.0 ,0.0
0.3 ,1.0
```

The above defined coordinates signify that starting from time "0" seconds the load will increase gradually from "0" to its maximum value of 3.0 mm at 0.3 seconds.

## VIII RESULTS AND CONCLUSIONS

We would be comparing the different output parameters for the material cards created namely Elastic, Elastic-Plastic and Elastic-Plastic with Fiber orientation and Elastic Plastic with damage. First we would be comparing the von-mises stress for the created material cards. As can be seen from figure 10 comparison of von-mises for different material cards were made.

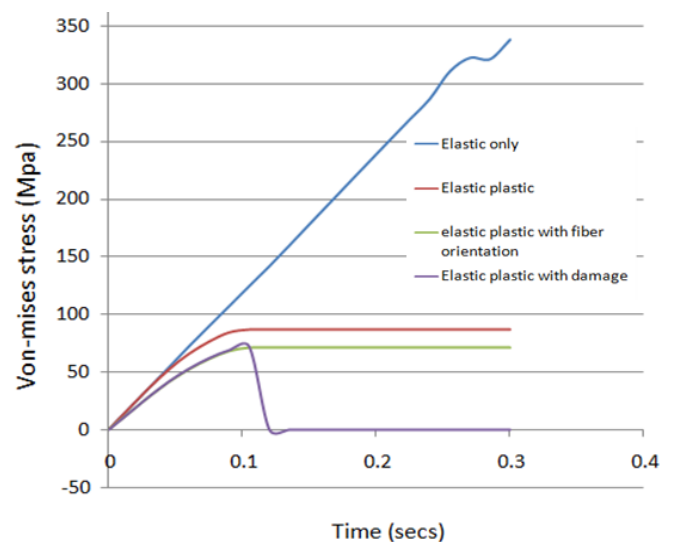


Fig.10 Comparison of Von-mises Stress versus Time for different material models

When we have defined "Elastic material card" only, we can see that there is a linear relationship between von-mises stress and time i.e. the von-mises stress increases gradually with a constant slope. Also it can be observed that the "Elastic plastic card" has more stress bearing capacity than "Elastic -plastic with fiber orientation". The maximum von-mises stress in "Elastic plastic card" is 86.2997 Mpa whereas for "Elastic -plastic with fiber orientation" is 71.49 Mpa. This is due to the fact that the "Elastic -plastic with fiber orientation" is the averaged material card that is it takes into account the effect of fiber orientation. That is it is average of material properties in parallel as well as perpendicular direction. Hence its material properties are somewhat weak as compared to "Elastic plastic card." The

“Elastic plastic card” has material properties for along the fiber orientation and it has more stress bearing capacity.

Now considering the behavior of “Elastic-plastic with damage” we can observe that at a time step of 0.12 seconds fracture happens. We have defined damage initiation criteria which consists of the fracture strain and stress triaxiality parameters. Once the “Equivalent plastic strain-PEEQ” in the model reaches the fracture strain value corresponding to that state of stress, the fracture happens. As in our case we have defined a fracture strain of 0.03 for tensile stress state .Now at a timestep of 0.12 seconds the PEEQ for some elements exceeds the fracture strain defined for tensile state stress. Hence at this timestep fracture happens and stress becomes equal to zero.

We can say that the time step 0.12 seconds is the critical step at which the damage occurs. The von-mises contour plots of "elastic plastic with fiber orientation" and "elastic plastic with damage" are plotted at this time step and are shown in the subsequent figures 11 and figure 12 respectively:

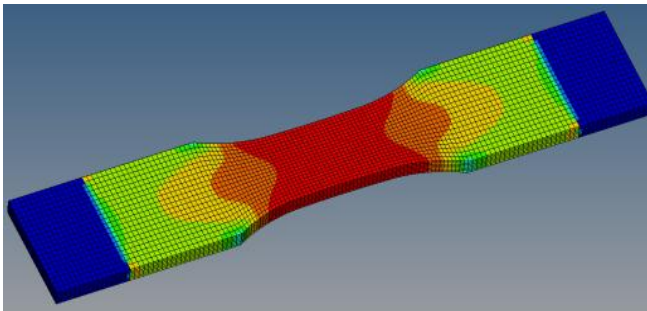


Fig 11 Contour plot of von-mises stress for an “elastic plastic with fiber orientation” material card at time 0.12 secs

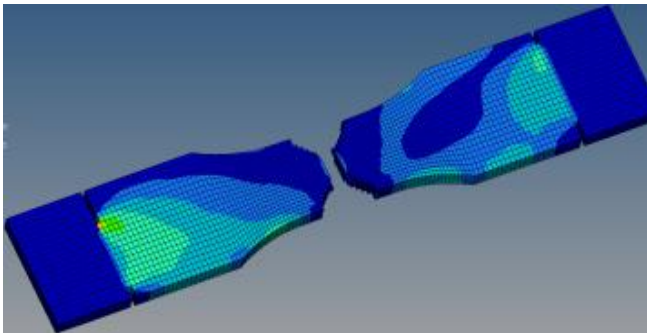


Fig 12 Contour plot of von-mises stress for an “elastic plastic with damage” material at time 0.12 secs

As can be observed from figure 11, there are red regions indicating that they are the most stressed one whereas in figure 12 the elements gets deleted when they satisfy the damage criteria. As can be seen from figure 12, the fracture happens at the center of the specimen and the most stressed elements get deleted. The element deletion depends upon the state of material that is stress triaxiality and fracture strain defined.

The figure 13 shows the comparison of stress triaxiality for considered cases of material. As can be seen, in case of “elastic material” card the stress triaxiality increases gradually with time and then remains constant. All the material cards have almost identical stress triaxiality upto time of 0.105 seconds. After that the stress triaxialities of “elastic plastic” and “elastic plastic with fiber orientation” increases gradually whereas in case of “elastic plastic with damage” the stress triaxiality becomes zero. This is due to the fact that PEEQ becomes more than fracture strain specified for that particular stress triaxiality value and subsequently damage occurs. Hence the material loses its all load bearing capacity.

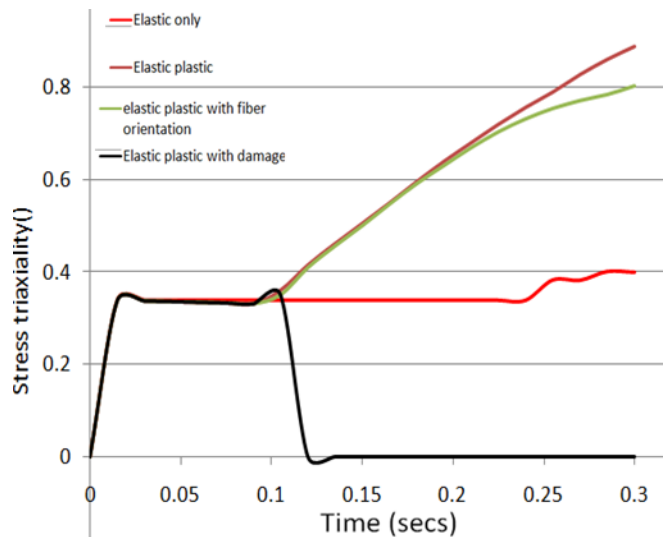


Fig. 13 Comparison of Stress triaxiality versus time of element 16278 for different material models

In case of damage material card the variable DUCTCRT tells us about whether damaged has reached or not. The DUCTCRT is ductile damage initiation criterion. When its value reaches unity then it means the damage has occurred. The below figure 14 shows the DUCTCRT with respect to time for the damage material card. As can be seen the value of DUCTCRT becomes equal to unity at time 0.12 seconds and here the material fracture occurs at this time step

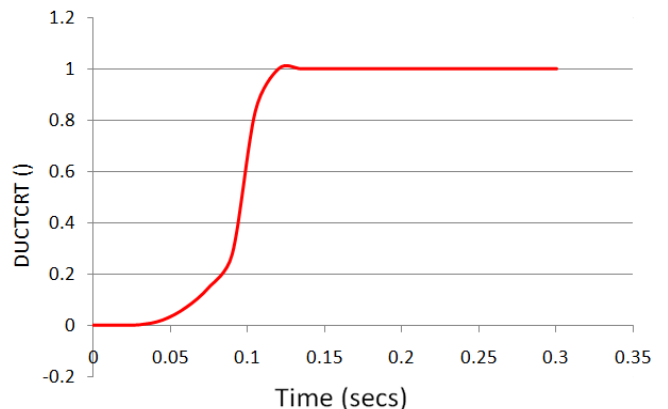


Figure 14 DUCTCRT values obtained with respect to time of Element 16278 for damaged material model

## REFERENCES

- [1] Nachketa Tiwari, "Introduction to composite material and structure," Indian institute of technology kanpur.
- [2] Prof.P.C.Pandey, "Composite material, Department of civil engineering", IISc banglore.
- [3] Sharifah Imihezi Syed Shaharuddin, "A Review of the Effect of Moulding Parameters on the Performance of Polymeric Composite Injection Moulding," Turkish J. Eng. Env. Sci. 30 (2006), 23 - 34 vol.
- [4] G. Maresca, P.P.milella, G.Pino, "A critical review of triaxiality based failure criteria," ANPA - Via V. Brancati, Roma
- [5] S.Kut, "A simple method to determine ductile fracture strain in a tensile test of plane specimen," ISSN 0543-5846 METABK 49(4) 295-299 (2010)
- [6] Henning lavenger, "Simulating Ductile Fracture in Steel using the Finite Element Method: Comparison of Two Models for Describing Local Instability due to Ductile Fracture," Faculty of mathematics and natural sciences, University of Oslo
- [7] Simulia Abaqus 6.14,"Abaqus user analysis guide",vol3 and vol4
- [8] Dr. Wolfgang Korte," Material Behaviour of Short-Fiber-Reinforced Plastic Parts ", PART Engineering GmbH
- [9] "Plastics determination of tensile properties", Part 1-General principles", ISO 527-1:2012(E)
- [10] "Plastics determination of tensile properties, Part 4 -Test conditions for isotropic and orthotropic fiber reinforced composite plastics", ISO 527-4: 1997(E)
- [11] Bryan Harris," Engineering composites material",The Institute of Materials, London 1999