

Ductile Fracture Prediction of Automotive Suspension Components

John George and Kishore Pydimarry
 Honda R&D Americas, Inc.

Jeremy Seidt and Kelton Rieske
 Ohio State University

ABSTRACT

Characterization of the plastic and ductile fracture behavior of a ferrous casting commonly used for the steering knuckle of an automotive suspension system is presented in this work. Ductile fracture testing for various coupon geometries was conducted to simulate a wide range of stress states. Failure data for the higher stress triaxiality were obtained from tension tests conducted on thin flat specimens, wide flat specimens and axisymmetric specimens with varying notch radii. The data for the lower triaxiality were generated from thin-walled tube specimens subjected to torsional loading and compression tests on cylindrical specimens. The failure envelopes for the material were developed utilizing the test data and finite element (FE) simulations of the corresponding test specimens. Experiments provided the load-displacement response and the location of fracture initiation. FE simulations were conducted to calculate all the stress states, Lode angles and strain components at the point of fracture initiation. Finally, comparisons of the predicted fracture load with data from physical tests are presented for a load case of the steering knuckle, which introduces complex stress states.

CITATION: George, J., Pydimarry, K., Seidt, J., and Rieske, K., "Ductile Fracture Prediction of Automotive Suspension Components," *SAE Int. J. Engines* 10(2):2017, doi:10.4271/2017-01-0318.

INTRODUCTION

Reliable prediction of ductile fracture behavior of metals is very valuable during the design of mechanical systems. Equivalent strain to fracture is commonly used as a measure to define the material ductility. Numerous researchers [1,2] have demonstrated that the failure strain of the material is not constant and it changes significantly with the state of strain of the material undergoing complex loading conditions. The most important parameter controlling the fracture of materials is the stress triaxiality. Stress triaxiality, η , is defined as the ratio of the hydrostatic pressure, σ_H , to the von Mises stress, $\bar{\sigma}$ (1) [3].

$$\eta = -\frac{\sigma_H}{\bar{\sigma}} \quad (1)$$

Other studies [4,5] have discovered that Lode parameter is also a significant variable in fracture characterization, especially at low levels of stress triaxiality. Lode parameter, μ , is defined as (2) [3]

$$\mu = \left(\frac{I_3}{\bar{\sigma}^3}\right)^3 \quad (2)$$

where I_3 is the third invariant of the stress tensor. The fracture model thus becomes a function of both triaxiality and Lode parameter (3).

$$e_p^f = f(\eta, \mu) \quad (3)$$

The goal of this work was to develop a fracture surface for a ferrous casting commonly used in the knuckle of an automotive suspension system. The coupon geometries that introduced various stress states and Lode parameters are presented. Tension tests were conducted on specimens for plane stress, axisymmetric and plane strain. Other stress states were achieved through the testing of torsional loading of thin-walled tube specimens and compression tests on cylindrical specimens.

The limiting fracture surface was obtained by correlating the experimental results with detailed numerical simulations utilizing the ABAQUS solver. Experiments provided the load-displacement response, the location of fracture initiation and the surface strains. FE simulations of the coupon geometries were conducted and correlated with the experimental results. The results from the correlated model were used to calculate the stress states and the limiting fracture envelope.

The fracture surface developed for the knuckle material was implemented in the ABAQUS solver. The material model was validated by comparing the results from simulations with test data for practical load case like the cornering load, which introduced complex stress states on the knuckle.



TEST SETUP & CAE MODEL

Test Setup

The fracture tests were performed using two load frames, an MTS 380 system with a load capacity of 250 kN and an Instron 1321 frame with a load capacity of 90 kN at a strain rate of 0.001 1/s. The 3D-digital image correlation (DIC) method was used to measure all the displacement and strain fields. The test setup used to measure the strain and displacement fields is shown in [Figure 1](#). As shown in [Figure 1](#), two digitally-aligned and time-synchronized cameras were used to collect the data. The data were processed using the VIC-3D™ software platform.



Figure 1. Test setup

Material Data

The first part of the study was to determine the material data required for the CAE simulations. It is well known that ferrous castings show different behavior in tension and compression [6]. Both tension and compression tests were conducted to generate the true stress-strain data required for the ABAQUS solver. The specimen geometry that was used to generate the material data in tension and compression is shown in [Figure 2](#) (all dimensions are in mm).

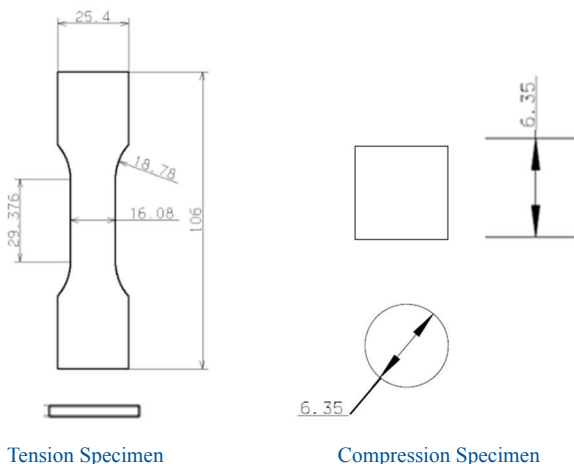


Figure 2. Coupon geometry used for the tension and compression test

The curves and the surface strain plots obtained from the tests are shown in [Figure 3](#). As observed in [Figure 3](#), there was a significant difference in the elastic-plastic response of the material under tension and compression. The true stress and plastic strain information from both curves was used in conjunction with the “CAST IRON PLASTICITY” model [3] available in ABAQUS solver to model this material behavior.

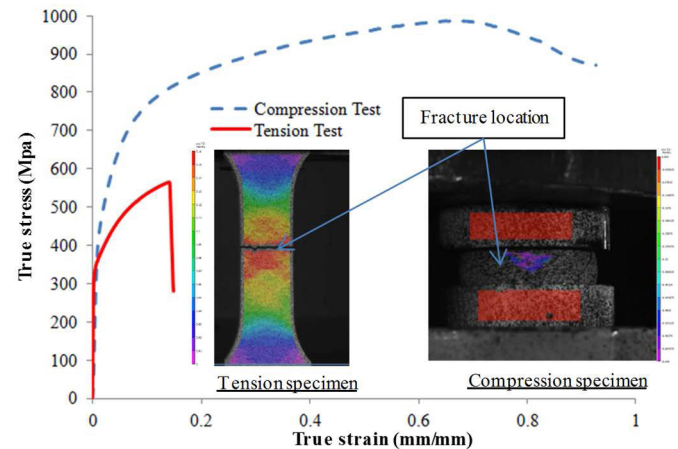


Figure 3. True stress-strain curves from tension and compression tests

FE Model

The FE mesh of all the test coupons was generated using fully-integrated brick elements (C3D8I) of the ABAQUS solver. An edge length of 0.5 mm was used to generate the mesh of all the test coupons. The specimens were held fixed at the gripping locations on one end and displacement was applied at the other end as it was done during the test. The force-displacement curves were generated using the section force at the minimum cross section of the specimen. The FE model is shown in [Figure 4](#).

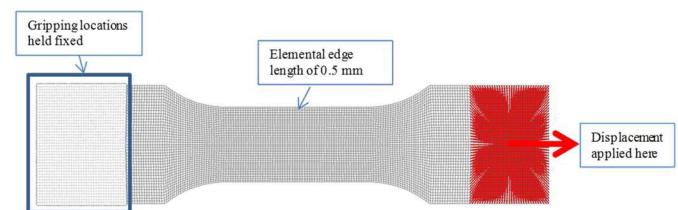


Figure 4. FE model and boundary conditions

COUPON TESTS

Tension, torsion and compression tests were conducted on various coupon geometries designed to provide a wide range of triaxialities and Lode parameters in order to sufficiently populate the fracture surface. Tension tests typically cover triaxialities of 0.33 to 1.0, torsion tests provide triaxiality values around 0.0 and compression tests account for triaxialities below -0.33. All the coupons used in this study were machined from a cast block of 175 mm x 30 mm x 30 mm. The results from each set of tests are described below.

Tension Tests

Tension tests were conducted for axisymmetric specimens, plane stress specimens and plane strain specimens to obtain various stress states at the failure location. The geometries of the tensile specimens are shown in Figure 5. The specimens with the larger radii have the stress state corresponding to uniaxial tension and provided triaxialities in the 0.3 to 0.5 range, while the specimens with tighter radii provided plane strain conditions with higher triaxiality ranges from 0.5 to 1.0.

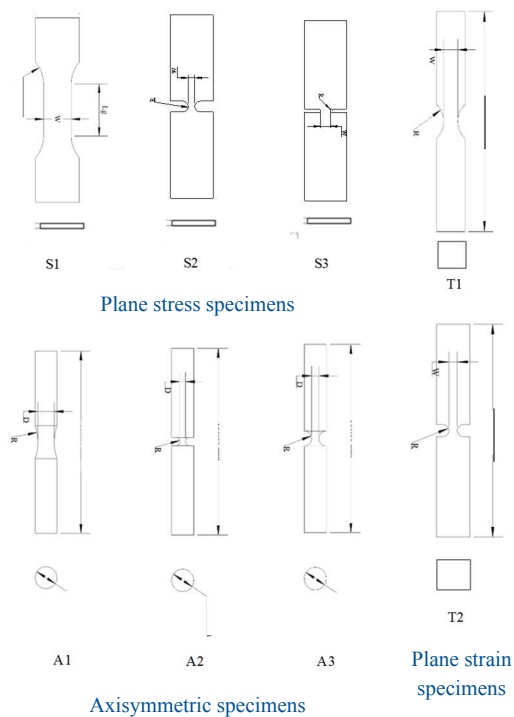


Figure 5. Geometry of the specimens used for tension tests

The load-displacement curves obtained for the tensile test specimen “S1” are shown in Figure 6. The test showed a sudden drop in load in the force-displacement response during the initiation of fracture. Similar sudden drops in load were observed in all the tensile specimens. The displacement at the sudden drop in load was considered as the fracture initiation point for all tests. Due to the abrupt propagation of the fracture, it was very difficult to determine where the fracture initiated. However, previous studies have shown that the fracture initiated at the center of the tensile specimens [7]. Therefore, the plastic strains and stress states at the center of the specimens were used to develop the limiting fracture surface.

Numerical simulations of all the specimens were carried out to predict the load displacement curves, utilizing the material model and the modeling method described in the previous section. The force-displacement data from the test, as well as the surface strain levels at the fracture initiation load from the DIC, were correlated with the results from the CAE model. The displacement where the sudden drop in load is observed during the test was used as the reference

point to calculate the fracture strain at the centroidal element from the CAE model. The correlation for the force-displacement curve and the fracture strain for the S1 specimen is shown in Figure 7. The surface strain fields at fracture initiation are plotted in Figure 8. The results show that force at fracture (error within 1%), as well as the strain levels, correlates very well with the test for the S1 specimen.

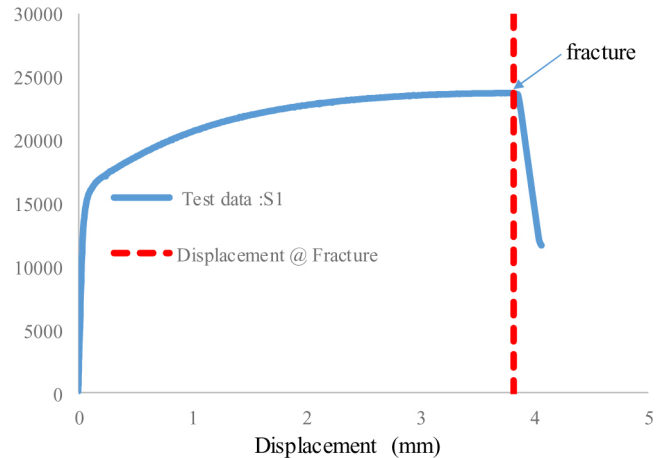


Figure 6. Load-displacement curves from tensile tests

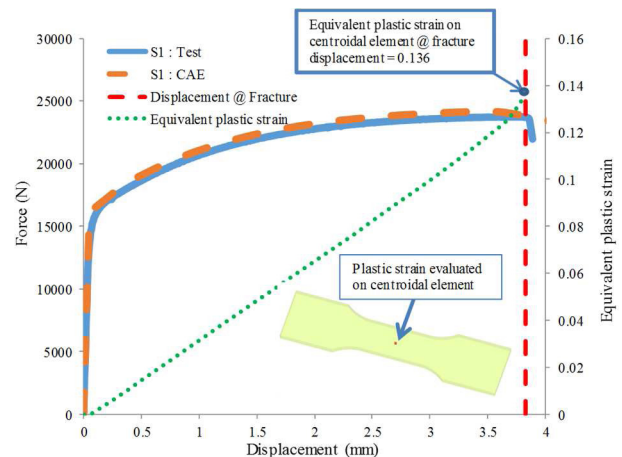
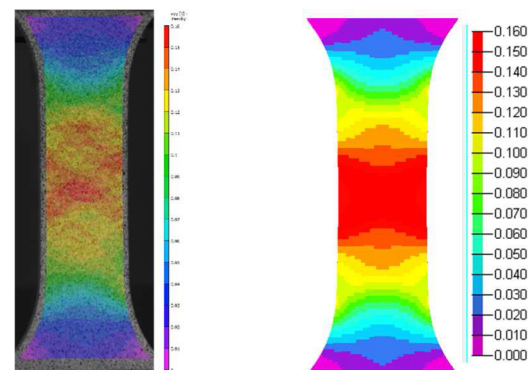


Figure 7. Load-displacement curves and equivalent plastic strain plot: S1 specimen



DIC measurements

CAE Results

Figure 8. Surface strain field comparison at fracture initiation: S1 Specimen