Visit the Resource Center for more SIMULIA customer papers

Pipeline Rupture in Abaqus/Standard with Ductile Failure Initiation

Abaqus Technology Brief

Summary

Defects may be introduced in metal pipelines during construction, repair, or by accident. A common example is a gouge from a backhoe bucket or other heavy equipment. At the site of a local defect operating stresses may become sufficiently concentrated to induce plastic deformation and material damage, possibly resulting in eventual failure of the pipeline.

Historically, methods for assessing the structural integrity of a damaged pipe have been based on experimental tests. The Abaqus finite element suite includes the ability to simulate the initiation and evolution of damage in metals, providing a low-cost alternative to laboratory structural testing. In this Technology Brief, Abaqus/Standard will be used to predict the burst pressure of a steel pipeline with a notch-type defect. A ductile damage initiation criterion is used, and favorable comparison with available experimental data will be shown.

Key Abaqus Features and Benefits

- Damage initiation and failure modeling for ductile metals
    - Ductile and shear initiation criteria allow for the modeling of two primary fracture mechanisms: coalescence of voids and shear banding
    - Can be used for bulk or sheet metal analyses
    - Available for all mechanical elements and allows for element removal

Background

Pipelines are a critical component of industrial infrastructure and are used world-wide to transport liquids and gases. During the course of its lifetime, a metal pipeline may sustain mechanical damage in such forms as dents, gouges, or weld defects. When damage is detected, a decision to monitor, repair, or replace is necessary.

A body of assessment guidelines for determining the fitness-for-purpose of a damaged pipeline has been built over the past several decades. As discussed in [1], many of these methods rely on experimental results and semi-empirical procedures; as such, their validity may be limited when considering loadings, materials, or specific damage configurations that are outside the scope of their assumptions.

With the ability to include the effects of damage initiation and evolution in the analysis of a ductile metal pipeline, the Abaqus finite element suite can complement existing methodologies by adding a more general predictive capability. Specifically, two types of damage initiation criterion are offered: ductile, based on the nucleation, growth, and coalescence of voids, and shear, based on shear band localization. We will focus on the use of the ductile criterion. The present analysis will consider an internally pressurized pipe of API X65 steel with a gouge defect.

Finite Element Analysis Approach

The geometry of the model under consideration is shown in Figure 1. A simulated gouge, 100mm long, was introduced into the pipeline. A quarter-symmetric mesh of second order hex elements was generated, and internal pressure loading was applied. End forces were applied to simulate experimental closed end conditions, and the loads were increased linearly with time. In general, the specification of damage initiation is included in the material definition and must be used in conjunction with a plasticity model. In this analysis we use the Mises plasticity formulation. The mesh is shown in Figure 2.

The ductile damage initiation criterion is a phenomenological model. It is included in the analysis by specifying

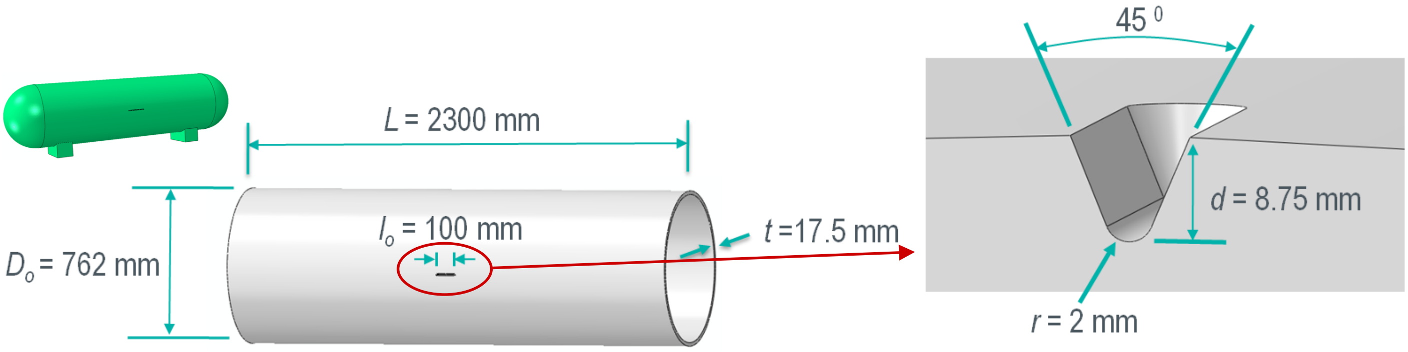

Figure 1: Geometry of damaged pipe model, with detail of simulated gouge

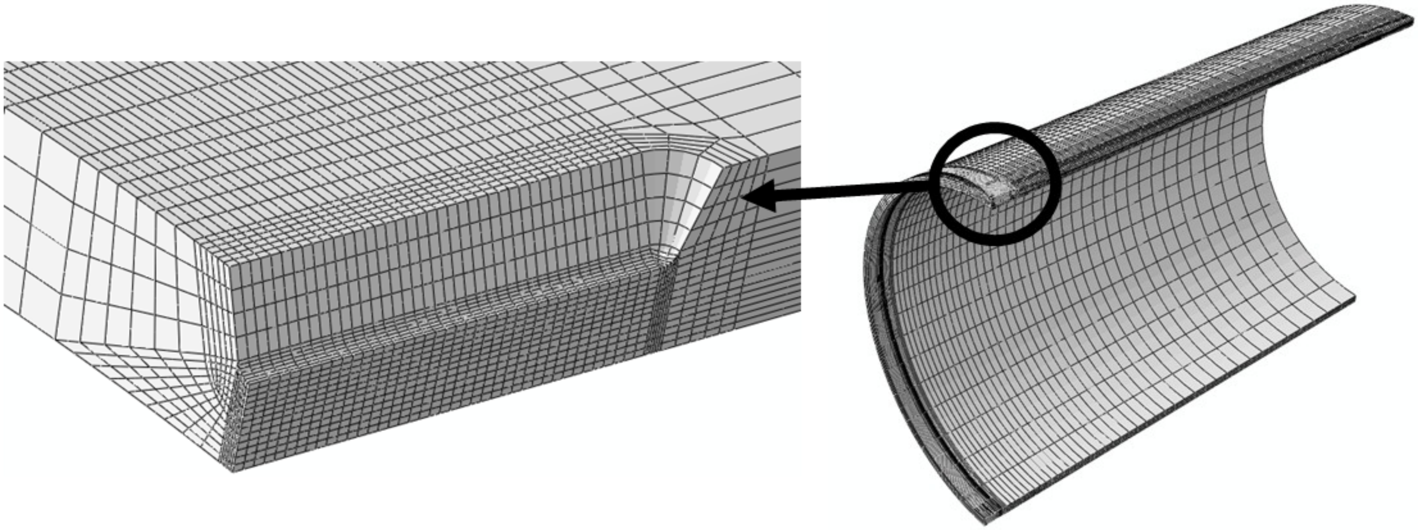

Figure 2: Quarter symmetric mesh of the pipe, with close-up of detailed mesh at the gouge

the equivalent plastic strain at damage initiation as a function of stress triaxiality and equivalent plastic strain rate. Stress triaxiality is defined as $\eta = -p/q$ (where p is the pressure stress and q is the Mises equivalent stress), and is known to play a role in damage growth. The procedure for calibrating the model used here is outlined in Appendix A.

Results

We compare the Abaqus/Standard results to the full scale experimental burst test data collected by Oh et al [2]. Axial and hoop strain measurements from two gauges are plotted against the analytical result in Figure 3. The gauges are located adjacent to the gouge, at circumferential distances of 24.9 and 54.9 mm. Favorable comparison with the experimental data is achieved.

A contour plot of the damage initiation output variable DUCTCRT is shown in Figure 4. Damage has initiated when this variable is greater than 1.0. From the contour,

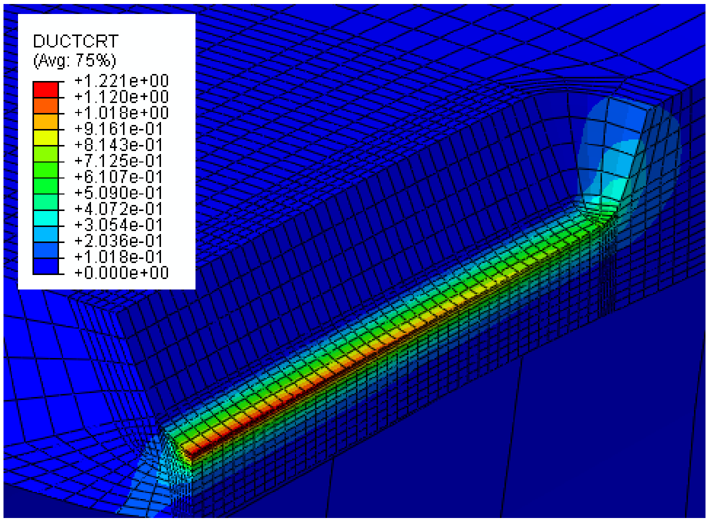

Figure 4: Damage initiation criterion in the pipe gouge

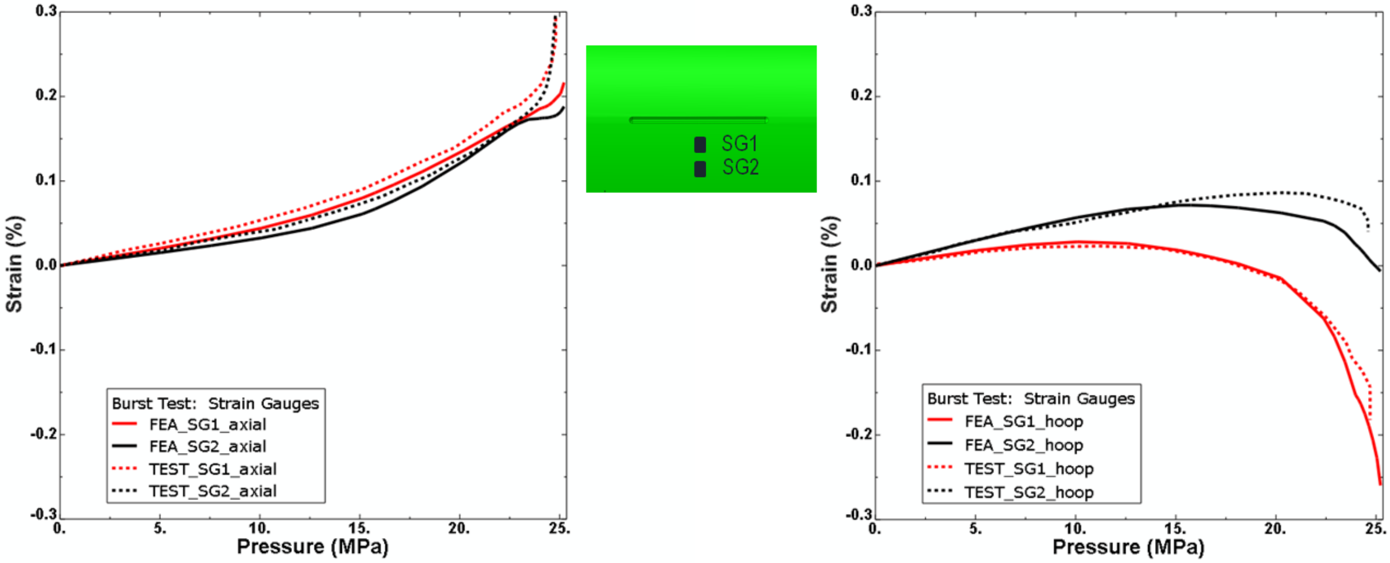

Figure 3: Experimental and analytical strain results near the notch and location of strain gauges

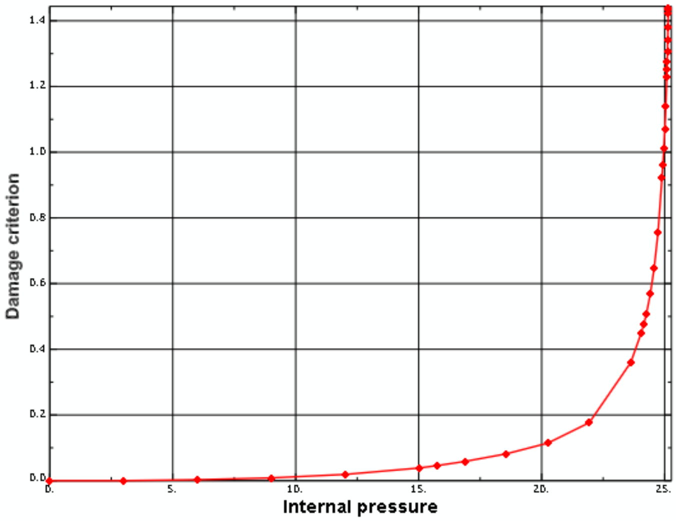

Figure 5: Damage criterion v. internal pressure load in critical element

we see that the critical element in the structure is in the root of the notch at the intersection of the symmetry planes. By X-Y plotting the initiation criterion in the critical element, a more precise determination of the failure pressure can be determined. In Figure 5, DUCTCRT at the centroid of the critical element is plotted against the applied internal pressure, and the threshold of 1.0 is crossed at a pressure of 24.97 MPa. The experimentally determined burst pressure 24.68 MPa.

The Abaqus prediction is compared to several other commonly used failure criteria in Table 1. The relatively good performance of the peak criteria is attributed to the same relative triaxiality of the smooth round tensile bar (~0.65) used to determine material properties as compared to that of the actual application (~0.6).

Table 1: Comparison of failure criteria

Criterion	Burst Pressure Predicted/ Experimental
Abaqus Ductile Failure	1.01
Net Section Collapse	1.04
Peak Max Principal Stress	1.01
Peak Equivalent Plastic Strain	1.01
Axial Crack (ASME B&PV Code Sec. XI)	0.86

Conclusions

In this Technology Brief we have demonstrated the utility of the Abaqus/Standard ductile failure initiation criterion for predicting the burst pressure of pipes with notch-type defects. Good correlation with available full scale experimental data has been shown. The Abaqus damage initiation and evolution capability for metals provides a general numerical tool that can supplement existing failure prediction methods that are based on empirical data.

References

1. Cosham, A. and Hopkins, P., "An overview of the pipeline defect assessment manual (PDAM)," 4th International Pipeline Technology Conference, 2004.
2. Oh, C.-K., Kim, Y.-J., Baek, J.-H., Kim, Y.-P., and Kim, W.-S., "Ductile failure analysis of API X65 pipes with notch-type defects using a local fracture criterion," International Journal of Pressure Vessels and Piping, Vol. 84, pp. 512-525.

Abaqus References

For additional information on the Abaqus capabilities referred to in this document please see the following Abaqus 6.12 documentation references:

- Analysis User's Manual
  - "Damage and failure for ductile metals," Section 24.2.1

About SIMULIA

SIMULIA is the Dassault Systèmes brand that delivers a scalable portfolio of Realistic Simulation solutions including the Abaqus product suite for Unified Finite Element Analysis, multiphysics solutions for insight into challenging engineering problems, and lifecycle management solutions for managing simulation data, processes, and intellectual property. By building on established technology, respected quality, and superior customer service, SIMULIA makes realistic simulation an integral business practice that improves product performance, reduces physical prototypes, and drives innovation. Headquartered in Providence, RI, USA, with R&D centers in Providence and in Suresnes, France, SIMULIA provides sales, services, and support through a global network of over 30 regional offices and distributors.

Appendix A - Determination of the Ductile Failure Initiation Parameters

To use the ductile failure initiation criterion in Abaqus/Standard, one must specify the equivalent plastic strain at damage initiation as a function of stress triaxiality and strain rate. As outlined in [2] this requires an experimental program, a finite element analysis of each test, and the construction of a failure loci.

Experimental tests were performed on round, notched bars. The notched specimens were used to capture the effect of stress triaxiality on yield and tensile strength. A schematic diagram of the bar geometry is shown in Figure A1. The bars had an outer diameter of 17.5 mm and length of 130 mm. Smooth bars and those with three different notch radii were tested: 1.5, 3 and 6 mm. The bars were loaded in tension until complete fracture was achieved.

Figure A1: Schematic geometry of round, notched tensile test specimen

Each test had a corresponding axisymmetric finite element analysis. Second order, reduced integration elements (CAX8R) were used in a half-symmetric mesh. In the critical location, an element size of 0.15 mm x 0.15 mm was used. The Mises plasticity model was employed, and nonlinear geometric effects were included. A comparison of the computed stress-strain response and the experimental measurements is shown in Figure A2. The Abaqus/Standard analyses were run until crack initiation occurred in the experiment and good correlation was obtained. It can be seen that as notch radius decreased, yield and tensile strengths increased, but strain to failure decreased. This behavior is consistent with the increasing triaxiality of the stress state with decreasing notch radius.

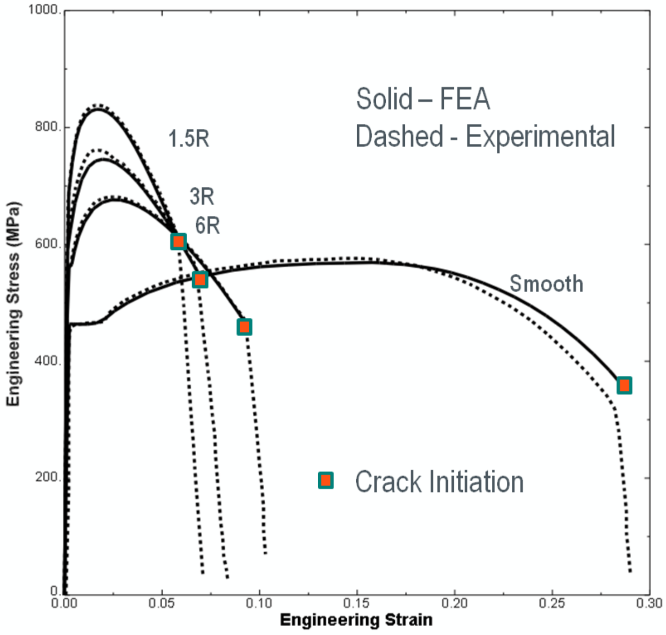

Figure A2: Experimental and Abaqus results for smooth and notched bar tensile tests

The critical location for each of the test specimens is at the center point of the bar. From the analysis results, the equivalent plastic strain as a function of the stress triaxiality at that location is plotted in Figure A3.

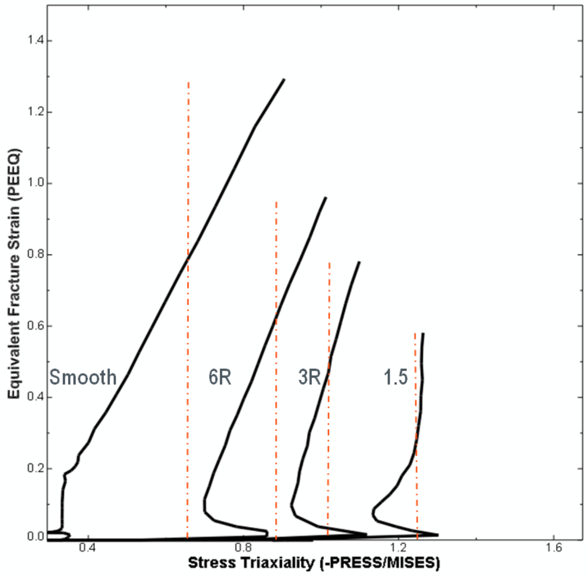

Figure A3: Equivalent strain vs. stress triaxiality, with values of average stress triaxiality

Included in Figure A3, shown by red dotted lines, is the average stress triaxiality for each specimen, defined as

$$\left(\frac{\sigma_{press}}{\sigma_{equiv}}\right)_{avg} = \frac{1}{\varepsilon_{ef}}\int_{0}^{\varepsilon_{ef}}\frac{\sigma_{press}}{\sigma_{equiv}}\,d\,\varepsilon_{equiv}$$

where ε_{ef} is the equivalent strain at failure initiation. Each equivalent strain trace in Figure A3 ends at its corresponding value of ε_{ef}. The points located by the black squares in Figure A4 are the equivalent stain—stress triaxiality data points used in the Abaqus ductile failure initiation definition. The red curve is the loci fitted by Oh et al. [2].

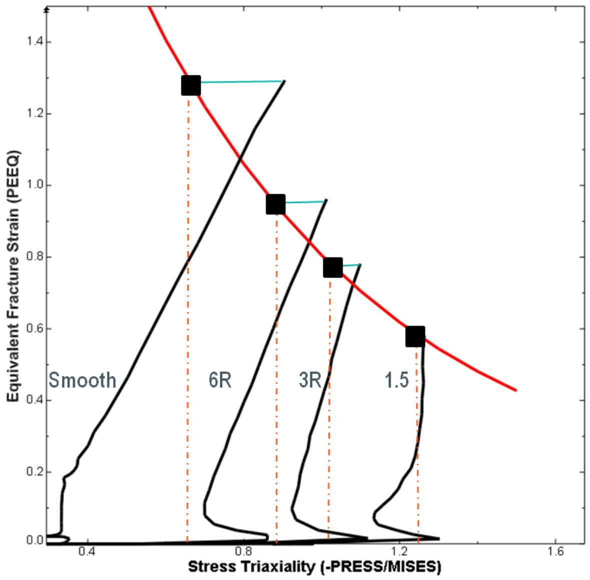

Figure A4: Equivalent strain—stress triaxiality pairings (black squares) used in the Abaqus ductile failure initiation criterion

DASSAULT SYSTEMES

Delivering Best-in-Class Products

CATIA Virtual Product Design	**EXALEAD** Information Intelligence
SOLIDWORKS 3D for Professionals	**NETVIBES** Dashboard Intelligence
SIMULIA Realistic Simulation	**3DSWYM** Social Innovation
DELMIA Virtual Production	**3DVIA** Online 3D Lifelike Experiences
ENOVIA Global Collaborative Lifecycle Management	

Europe/Middle East/Africa
Dassault Systèmes
10, rue Marcel Dassault
CS 40501
78946 Vélizy-Villacoublay Cedex
France

Asia-Pacific
Dassault Systèmes
Pier City Shibaura Bldg 10F
3-18-1 Kaigan, Minato-Ku
Tokyo 108-002
Japan

Americas
Dassault Systèmes
175 Wyman Street
Waltham, Massachusetts
02451-1223
USA

Visit us at
3DS.COM/SIMULIA

Visit the Resource Center for more SIMULIA customer papers