Transactions of the Korean Nuclear Society Autumn Meeting
Gyeongju, Korea, October 25-26, 2012

Element-Size-Dependent Damage Model for Ductile Crack Growth Simulation of Degraded Pipe

Jong-Hyun Kim [a], Nak-Hyun Kim [a], Yun-Jae Kim [a*]
[a] *Dept.of Mech. Engr., Korea Univ., 1-5Ka, Anam-Dong, Sungbuk-Ku, Korea*
[*] *Corresponding author: kimy0308@korea.ac.kr*

1. Introduction

Performing full-scale tests is important in defect assessment of structures, but it is very difficult to reflect the complex geometries and loading conditions in practice in plant assessments. An efficient tool is therefore needed not only to design full-scale complex tests but also possibly to minimize the need to perform such tests. One possible tool is virtual testing using finite element (FE) damage analysis based on the local approach.

Recently, the authors proposed a simple FE method to simulate ductile failure based on a phenomenological stress-modified fracture strain model [1]. The method is not new in the sense that the stress-modified fracture strain model is based on the well-known concept that the fracture strain for ductile fracture strongly depends on the stress state [2]. An advantage of this model is that, for a given material, the stress-modified fracture strain model can be determined in a robust way using notched bar tensile test results.

Needless to say, virtual testing using FE damage analysis should be ultimately used to simulate failure of large-scale components such as full-scale pipe tests (possibly with long, stable crack growth). To achieve this, an issue related to a finite element size for damage analysis needs to be resolved. In FE damage analysis, an element size is an important parameter and should be chosen to reflect the material's length scale (such as void spacing for ductile fracture). In this respect, FE ductile fracture simulations are typically performed using element sizes in the order of 100μm (which is typical void spacing for structural steels). Using such small elements, crack growth simulation up to millimeter length scales could possible but longer crack growth simulation is problematic, due to numerical problems. Thus, to simulate long, stable ductile crack growth in full-scale cracked pipes, existing methods need to be modified to incorporate larger element sizes.

In this paper, an element-size-dependent damage model based on the stress-modified fracture strain model has been proposed to simulate failure of full-scale cracked pipes. The proposed method is then compared with the published experimental full-scale pipe test data in Degraded Piping Program[3]. The concept of the element-size-dependent damage model is explained. Simulated results are compared with experimental data of circumferential through-wall cracked pipe.

2. Experiment

The pipe test considered in this study was carried out as a part of the Degraded Piping Program (Experiment number 4131-7; Data Record Entry 1.1.1.13) [3]. The test specimen was fabricated from a 273.1mm nominal diameter carbon steel (SA333 Grade 6) pipe. The wall thickness of the pipe was 18.3mm. The test specimen had an idealized circumferential through-wall crack (initial crack length was 34.6% of the pipe circumference, i.e., 124.6 degrees) and was loaded in four-point bending. The crack tip was sharpened with a saw so that the notch-tip radius was approximately 0.1mm. The pipe experiment was conducted at quasi-static loading rate (approximately 2.54mm/min) at 288°C. Data of load, load-line displacement (LLD), center line electric potential, crack growth length (Δa) were recorded during the test.

Fig. 2 shows the total moment versus LLD, obtained from the pipe test. The moment at crack initiation was 112kN-m (for both crack tips). The maximum moment for this experiment was 155kN-m. The average crack growth from initiation and maximum load was approximately 8mm.

3. Finite Element Analysis

To calibrate the element-size-dependent damage model, experimental tensile and C(T) test results are compared with FE results, as described below. Firstly conventional 3-D, elastic-plastic FE analysis (without using the damage model) was performed to simulate smooth bar tensile test. Symmetric conditions were fully utilized and the first order solid elements (C3D8 within ABAQUS [6]) were used with the element size of about 0.1mm. The FE mesh is shown in Fig. 3. To incorporate possible large deformation in tensile testing, the large (nonlinear) geometry change option was chosen. To simulate ductile fracture, the stress-modified fracture strain model was used. The procedure of determining stress-modified fracture strain was shown in Ref.[4]

To determine the stress-modified fracture strain, more data points are needed, for instance, using notched bar tensile tests with different notch radii. When notched bar tensile tests are not available (as is the case in the present problem), we need to simplify the equation. By the theoretical value of the Prandtl field [5], a value at high stress triaxciality of σ_m/σ_e=2.5 is assumed. As our experience on various structural steels suggests that the fracture strain at σ_m/σ_e=2.5 ranges from 0.1 to 0.2, two fracture strain criteria using 0.1 and 0.2 σ_m/σ_e =2.5 were used as initial trails.

$$\text{Criteria 1: } \varepsilon_f = 1.64\exp\left(-1.5\frac{\sigma_m}{\sigma_e}\right)+0.061 \qquad (1)$$

$$\text{Criteria 2: } \varepsilon_f = 1.44\exp\left(-1.5\frac{\sigma_m}{\sigma_e}\right)+0.166 \qquad (2)$$

By comparing simulated FE results with the C(T) test results, the critical accumulated damage for cracking, ω_c, was calibrated, which is assumed to be a function of the element size shown in Fig. 1. After ω_c value is determined for a given element size, actual size of the finite element damage analysis using the size of the given element can be performed. When the accumulated damage becomes unity, $\omega=\sum\Delta\omega=\omega_c$, ductile failure is assumed locally and incremental crack growth is simulated using the technique described.

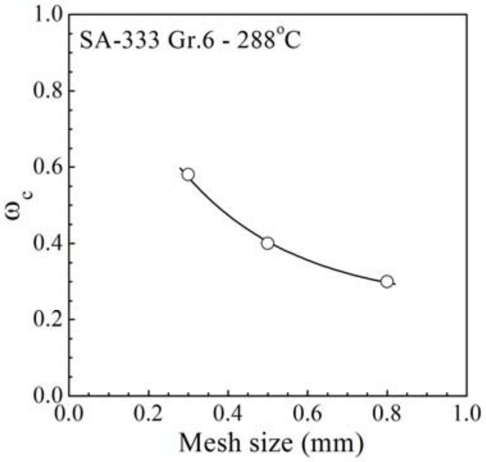

Fig. 1. Variation of the critical damage with an element size.

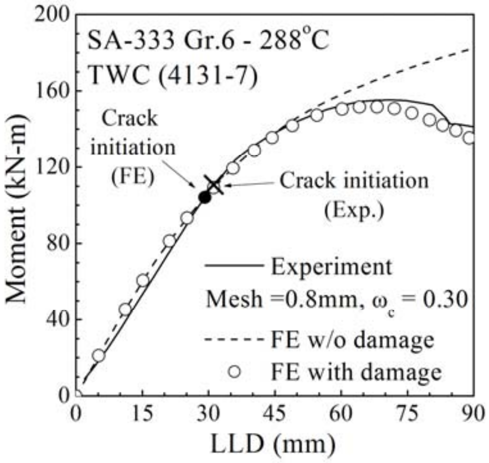

Fig. 2. Comparison of experimental data with simulated bending moment versus LLD curve.

The above failure simulation technique is implemented in the commercial FE program, ABAQUS[6] using the UHARD user subroutines. Also the element size and ω_c from C(T) specimen analysis were applied to full-scaled pipe test.

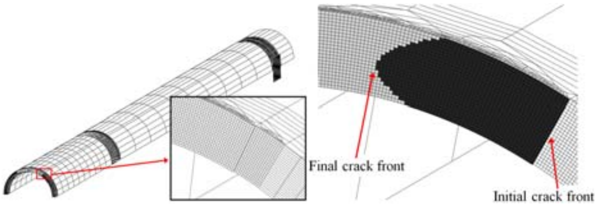

Fig. 3. FE mesh of cracked surface at LLD=90mm .

Simulated results are compared with experimental data in Fig. 2. For Moment-LLD data, conventional elastic-plastic FE results without damage analysis are also compared. Simulated FE Moment-LLD results with damage analysis agree very well with experimental data up to the pipe failure point. Fig. 3 shows the FE mesh after ~40mm of crack growth, where the darker area indicates the cracked area during loading.

4. Conclusions

In this paper, an element-size-dependent damage model is proposed to simulate long, stable ductile crack growth in structural components such as full-scale pipes. Simulated results agree well with experimental data.

REFERENCES

[1] Oh CS, Kim NH, Kim YJ, Baek JH, Kim YP, Kim WS., 2010, "A finite element ductile failure simulation method using stress-modified fracture strain model," Engng Fract Mech , Vol. 78, pp. 124~37.

[2] Rice JR, Tracey DM., 1969, "On the ductile enlargement of voids in triaxial stress fields," J Mech Phys Solids, Vol. 17, pp. 201~217..

[3] Pipe Fracture Encyclopedia, 1997, Vol.3, Pipe Fracture Test Data, Battelle.

[4] Kim NH, Oh CS, Kim YJ, Yoon KB, Ma YH., 2011, "Comparison of fracture strain based ductile failure simulation with experimental results," Int J Pres Ves Piping, Vol. 88(10), pp. 434~447.

[5] Ewing DJF, Hill R., 1967, "The plastic constraint of V-notched tension bars," J Mech Phys Solid , Vol. 15, pp. 115~124.

[6] ABAQUS Version 6.11. SIMULIA, "User's manual. Inc. and Dassault Systemes; 2011.