A COMPARISON OF GEOMETRICAL PARAMETER AND GURSON'S MODEL APPROACHES TO SIMULATIONS OF DUCTILE FRACTURE IN PIPELINES

Alberto Corigliano, Giulio Maier, Stefano Mariani[1]

In this paper ductile fracture processes are simulated by means of two different approaches. The first one is based on the use of the geometric parameters, namely Crack Tip Opening Displacement (CTOD) and Crack Tip Opening Angle (CTOA), which are assumed to govern the fracture process at the onset of crack propagation and during crack propagation, respectively. The second approach is based on the use of Gurson's porosity model. Finite elements simulations of crack processes in Three-Point-Bending (TPB) tests and in indented pressurized pipelines, carried out with both approaches, are compared with experimental results. A discussion on the applicability of both approaches is presented.

INTRODUCTION

In pressurized pipelines, especially in those for gas transportation, fracture processes are generally characterized by large scale yielding. Therefore elastic-plastic fracture mechanics provides the appropriate basis for assessments of safety against ruptures of buried pipelines. In such situations, the traditional J-integral approach to ductile fracture analysis becomes inadequate and, hence, alternative approaches have been proposed in the recent literature. The most popular approaches appear to be the one based on an extension of the J-integral theory by the introduction of a second parameter related to the triaxiality of the stress field ahead of the tip (see e.g. Nevalainen and Dodds (1)) and the approach resting on a phenomenological description of local damage processes. The latter methodology usually based on Gurson's model (Gurson (2)) which relates a fracture process to void initiation and growth up to coalescence, has been widely applied to simulations of crack advancements (see e.g. Xia at al. (3)).

[1] Department of Structural Engineering, Politecnico of Milano
P.za L. da Vinci 32, 20133 Milano, Italy

In this paper an analysis procedure is developed for numerical simulations of the crack process in an industrial perspective, centered on the use of *geometric parameters* (see Corigliano at al. (4), (5)), and it is compared to a procedure based on Gurson's model.

Fist, these two quite different approaches are comparatively discussed with reference to simulations of TPB tests. Second, the two procedures are examined and assessed for the evaluation of the ultimate pressure in indented damaged pipelines. The company Snam Spa provided financial support and experimental data to this study.

FRACTURE SIMULATIONS BASED ON GEOMETRIC PARAMETERS

In the approach based on geometric parameters, it is assumed that the crack starts propagating when the *CTOD* reaches a critical value, say $CTOD_c$. The identification of $CTOD_c$ can be carried out as suggested by the British Standards procedure (BS 5762) or according to a more refined modeling of the elastic-plastic behaviour in the process zone, as discussed in Corigliano at al. (5).

During the propagation phase, the crack growth behaviour can be predicted by means of the critical value of *CTOA*. Recent experimental measurements of $CTOA_c$ (Han at al. (6)) show the existence of a transient phase after the onset of crack propagation, where *quasi* steady-state crack propagation has not been reached yet. This early stage influences the global response during tests conducted on specimens made of high-toughness steel. In order to take into account this behaviour during finite element simulations, in Corigliano at al. (5) a function $CTOA_c(\Delta a)$ has been adopted, Δa being the crack advancement. $CTOA_c(\Delta a)$ starts with its highest value at the onset of crack propagation and reaches a constant lower value during quasi steady-state crack propagation (see Figure 1b). During the transient propagation phase, $CTOA_c$ is assumed to vary according to a parabola, which depends on three parameters: $CTOA_c^0$, the $CTOA_c$ value at crack initiation; $\overline{CTOA_c}$, the quasi steady-state $CTOA_c$ value; Δa_t, the crack advancement at which quasi steady-state crack propagation is reached. The identification of the above three parameters suggested in (5) is based on the minimization of a discrepancy norm between the experimental load-vs-displacement plot and a corresponding theoretical one based on a simplified model of the propagation phase.

Geometric parameters and finite element (FE) modelling have been adopted to simulate crack propagation through the ligament of single-edge notched specimens during TPB tests and through the thickness of an indented pipeline during the pressurization phase (see figure 3). In both situations, due to the symmetry, rectilinear mode-I crack propagation was assumed. The adopted FE structured meshes have a characteristic length l_e of finite elements in the fracture process zone. In 2D simulations of TPB tests and pressurized pipeline we have chosen l_e=1/50 of the specimen width and pipe thickness, respectively; in 3D simulation of the pipe l_e=1/20 of the pipe thickness. The materials considered are typical pressure-vessel structural steels described by means of Mises J_2 yield criterion and slight isotropic hardening with saturation.

During each simulation, *CTOD* and *CTOA* are computed on the basis of displacements of nodes lying along the crack flanks. The value of *CTOD* is obtained as the distance between two points located at the intersections between the deformed crack profiles and the straight lines emanating backward from the tip with a slope at 45° with respect to the propagation direction. When *CTOD* reaches its critical value the crack may start propagating and the node at the tip is released (i.e. is replaced by two separate nodes). Along the subsequent propagation phase, the value of *CTOA* is computed in the current configuration as the angle between the tangents to the deformed modelled crack profiles at the tip. At each step of the analysis, the *CTOA* is compared to the current critical value given by the function $CTOA_c(\Delta a)$. When $CTOA = CTOA_c(\Delta a)$ the tip node is released.

In the pipeline analyses, the crack propagates in the radial direction (through the thickness) along the longitudinal symmetry plane. The crack front is enforced to be straight and *CTOD* and *CTOA* are computed in the symmetry cross-section orthogonal to the propagation direction.

FRACTURE SIMULATIONS BASED ON GURSON'S MODEL

A complete simulation of the ductile fracture process from void initiation up to rupture is possible by making use of material models like Gurson's one (see e.g. Gurson (2), Xia et al. (3) and Tvergaard (7)). This type of models can be regarded as an elastic-plastic constitutive law with softening behaviour governed by the evolution law for the porosity f, i.e. by an internal variable susceptible to be interpreted as a measure, in an average way, of the void growth. Besides the elastic moduli (E, ν) and yield and ultimate stresses (σ_y, σ_u), the model includes parameters governing the influence of porosity on the material response, namely its initial value f_0 and the Tvergaard parameters q_1, q_2. In the present simulations f_0 is assumed to be zero, while q_1, q_2 are identified by a direct-search minimization of the discrepancy between numerical and experimental load-vs-displacement plots concerning the TPB tests.

Gurson's material model has been confined to a strip of elements (or *computational cells)* along the propagation direction, like in (3), all other FEs obeying a standard J_2 plasticity law, like in the simulations based on the geometric approach. The thickness of the aforementioned computational cells has been chosen in the range 200-300 μm, as suggested in (3) on the basis of micromechanical considerations (average distance among largest inclusions). In a sense, this can be regarded as a simple *regularization* provision in the presence of constitutive instability due to softening. To simulate extensive crack growth, the node release technique is adopted also within this approach. The current tip node is released when the porosity f reaches the *extinction value* f_E=0.2, which can be interpreted as a critical threshold at which the material is no more able to sustain tractions.

SIMULATIONS OF THREE POINT BENDING TESTS

The TPB tests simulated concern specimens, labelled 1 and 2, made of steels with different fracture toughness and equal elastic moduli (E=207000 MPa, ν=0.3). Their geometric and material data are collected in Table 1 together with the values of the fracture parameters identified by the aforementioned procedures. Here B is the specimen thickness, S the initial span and a_0 the initial crack length. The values of the Tvergaard parameters employed in FE simulations are: TPB 1: q_1=2, q_2=1.68; TPB 2: q_1=1.5, q_2=1. Two-dimensional simulations have been carried out by both the methods described in previous Sections. Plane-strain hypothesis has been adopted in view of the square section of the specimens. The mesh is composed of 989 four-node isoparametric elements (see Figure 1a). Large strains and displacements have been assumed.

In Figures 2 experimental load (P) vs displacement (u) plots are compared to numerical results obtained by the geometric parameter approach and by Gurson's model approach. As far as TBP 2 is concerned, the geometric approach turns out to predict a significantly lower response during quasi steady-state crack growth. Better agreement with experiments obtained by Gurson's model is due to the identification of parameters q_1, q_2 based on these data.

In order to validate the simplified identification procedure of the relationship $CTOA_c=CTOA_c(\Delta a)$, in Figure 1b the functions thus obtained are compared to that computed during the simulations based on Gurson's model when the current tip node is released. A fairly good agreement can be observed.

Specimen	$B=W=S/4$ *(mm)*	a_0 *(mm)*	σ_y (N/mm^2)	σ_u (N/mm^2)	$CTOD_c$ *(micron)*	$CTOA_c^0$ *(degrees)*	$\overline{CTOA_c}$ *(degrees)*	Δa_t *(mm)*
1	12	6.26	482	565	33.52	25	5.25	1.99
2	14	5.33	553	605	56	180	40	1.92

TABLE 1 - TPB test data.

SIMULATIONS OF FRACTURE IN AN INDENTED PRESSURIZED PIPE

The assumption of crack propagation along the thickness of an indented-pressurized pipe is fairly realistic and often legitimate for real-life situations where indentation and notch have been caused e.g. by an agricultural tool hitting the pipeline; crack propagation may occur due to the internal pressure in the pipe.

Figure 3 represents a segment of pipeline with a longitudinal notch. First, the structure is subjected to a radial inward load which causes an indentation around the defect. This loading phase (α) leads to a residual deformed configuration in which the diameter of the pipe below the defect is reduced by 3% of its original length. A second phase (β) consists of an internal pressurization which leads to crack propagation along the thickness.

The above two phases (α) and (β) of the loading process have been simulated by means of both plane strain (2D) and three-dimensional (3D) finite element discretizations in space. The spatial discretization, in the former (2D) case has been performed by four-node isoparametric elements, in the latter (3D) case by shell elements far from the defect zone and brick elements around it.

In the 2D interpretation of the system, the analyses for the phase (α) have been carried out assuming a traditional Mises elastic-plastic model over the whole domain and Gurson's model in the strip of elements along the ligament on the radial direction below the notch. The final states of the above analyses have been used as starting states for the numerical simulations of crack propagation phase (β): the former by means of the geometric parameter method with parameters identified on the basis of a TPB test; the latter by means of Gurson's model. For the 3D simulation of phase (α) a J_2 (von Mises) plasticity model was adopted over the whole mesh, while phase (β) has been simulated only by means of the geometric parameter method.

The computed response during phase (β), driven by increasing internal pressure, is represented in Figure 4 and compared to experimental data, through the relationship between the internal pressure (p) and the displacements (s) of the point lying on the internal surface of the pipe just below the defect. The 2D simulations exhibit more compliance than the 3D one. Clearly, this difference is primarily due to the fact that in 2D the contribution to stiffness and resistance provided by the non-indented sections around the damaged zone cannot be taken into account. The difference between the experimental plot and the 3D-simulated one arises mainly from the assumption of straight crack front in the 3D simulations.

In all the analyses the internal pressure has been the control variable, so that its value at the last converged time-step solution can be regarded as failure pressure.

The pipeline crack propagation analyses governed by the geometric parameters led to failure pressures about 70% lower with respect to the non-propagation case. The ultimate values of pressure at the end of the propagation phase are not very different in 2D and 3D interpretations, mainly because, after the crack propagation along its thickness, the pipe recovers almost completely the original cylindrical shape. The Gurson's model approach predicts a lower failure pressure due to earlier lack of convergence during the analyses.

CLOSURE

Finite element simulations of ductile fracture processes in pressure-vessel steels have been presented, based both on geometric parameters and Gurson's model. The results of this study can be considered as a positive test for the applicability of the geometric approach employing critical values of fracture parameters obtained from the identification procedures mentioned in the second Section. The method appears to be suitable only for geometries and loading conditions which lead to mode I rectilinear crack growth, like in the pipeline considered herein.

Gurson's model approach is more versatile, but computationally more costly and it requires the calibration of the characteristic dimension l_e of the computational cells and the identification of the parameters concerning the evolution of the porosity.

REFERENCES

(1) Nevalainen, M. and Dodds, R. H, Int. J. Fracture, Vol. 74, 1995, pp. 131-161.

(2) Gurson, A. L., J. Engng. Mat. Techn., Vol. 99, 1977, pp. 2-15.

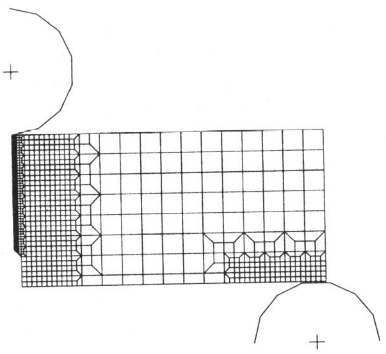
(3) Xia, L., Shih, C. F., and Hutchinson, J. W. , J. Mech. Phys. Solids, Vol. 43, 1995, pp. 389-413.

(4) Corigliano, A., Maier, G., and Mariani, S., Proceedings COMPLAS V, Barcelona, 1997, pp. 1033-1038.

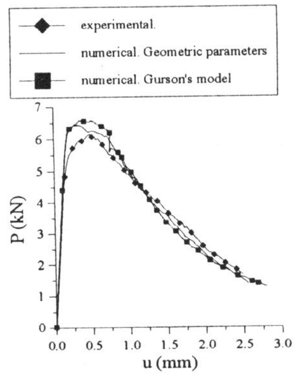
(5) Corigliano, A., Maier, G., and Mariani, S., Engineering Structures, 1998, to appear.

(6) Han, G., Sutton, M. A., Chao, Y. J. and Lyons, J. S., Engng. Fracture Mech., Vol. 52. 1995, pp. 525-555.

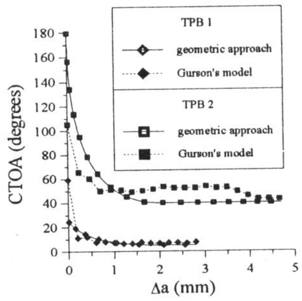
(7) Tvergaard, V., Adv. Appl. Mech., Vol. 27, 1990, pp. 83-151.



(a)

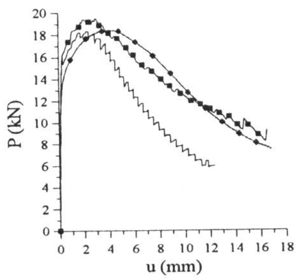


(a)



(b)

Figure 1. (a) FE mesh for the TPB test.
(b) Functions $CTOA_c = CTOA_c(\Delta a)$.



(b)

Figure 2. TPB: load-vs-displacement plots
(a) Specimen 1. (b) Specimen 2.

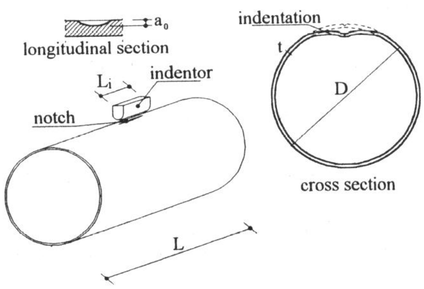


Figure 3. Geometry of the tested pipeline.

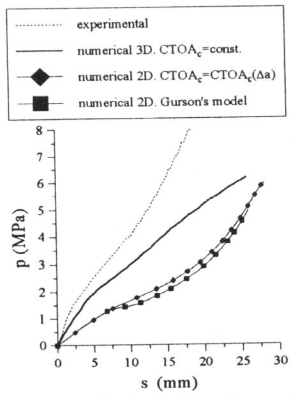


Figure 4. Pressurized pipeline: internal pressure-vs-displacement plot.