A HYBRID FINITE-ELEMENT SIMULATION
OF SOLID FRACTURE

ALEXANDER V. POTAPOV
Department of Mechanical Engineering, University of Southern California
Los Angeles, California 90089-1453, USA

CHARLES S. CAMPBELL
Department of Mechanical Engineering, University of Southern California
Los Angeles, California 90089-1453, USA

Received 2 April 1996

This paper describes an extension to a computer simulation of solid fracture. In the
original model, rigid elements are assembled into a simulated solid by "gluing" the ele-
ments together with compliant boundaries which fracture when the tensile strength of
the glued joints is exceeded. The current extension applies portions of the finite el-
technique to allow changes in the shapes of elements. This is implemented at the element
level and no global stiffness matrix is assembled; instead, the elements interact across
the same compliant boundaries used in the rigid element simulation. As a result, the
simulated material can conform to any desired shape and thus can handle large elastic
and plastic deformation. This model is intended to study the propagation of multitudi-
nous cracks through simulated solids to aid the understanding of problems such as the
impact-induced fragmentation of particles.

Keywords: Fracture; Fragmentation; Computer Simulation.

1. Introduction and Previous Work

In Refs. 1–3, a computer simulation for solid fracture was proposed. The essence
of that model was to compose a breakable solid body out of small unbreakable and
nondeformable polygonal elements "glued" together by compliant joints. The glued
joints may be thought of as a collection of fibers that connect initially coincident
points on the edges of neighboring elements. Figure 1 shows schematically how the
applied forces are resisted by stretching the fibers that make up the joint. (The fibers
are stretched by microscopic separation and overlap of the element surfaces, which is
shown greatly exaggerated here.) Each fiber has stiffness $K_n$ in the direction normal
to the joint, such that if $\vec{s}$ is a vector along the direction of the stretched fiber and
$\vec{n}$ is a unit vector normal to the joint, $K_n \vec{n} \cdot \vec{s}$ is the force per unit joint length
acting in the $\vec{n}$ direction. Similarly, the fibers have a stiffness $K_t$ in the direction
tangential to the joint so that, if $\vec{t}$ is a unit vector in the tangential direction, the
tangential force per unit length is $K_t \vec{t} \cdot \vec{s}$. Each fiber can withstand any compressive
normal force, but can only withstand tensile forces up to specified limit $\sigma_{\text{tens}}$; if the tensile strength is exceeded, the corresponding part of the joint breaks and can no longer withstand tensile forces. The free (i.e., not-glued) element boundaries and those portions of the glued boundaries that have broken can experience "collisional" contacts with other free boundaries. The methods for handling collisional contacts have their origins in simulation techniques for granular flows (see the review in Ref. 4) and are described in detail in Ref. 5. In such simulations, the particles are allowed to overlap slightly and produce a resistance that is proportional to the overlap; thus collisional contacts can only withstand compressive forces. As shown schematically in Fig. 2, the force $\mathbf{F}$ generated by collisional contacts is decomposed into normal and tangential components (i.e., in the $\mathbf{n}$ and $\mathbf{t}$ directions) relative to the orientation of the force with respect to a "contact line". (The concept of the "contact line" is described in detail in Refs. 1-3.) In this case, the force $F_n$, that acts normal to the contact line, is proportional to the area of overlap. It can be seen that the "fiber" concept, used for glued joints, also produces a compressive force that is proportional to the area of overlap, hence the same proportionality constant, $K_n$, should be used for collisional contacts so that the force is continuous across a glued/collisional transition. The force tangential to the contacting surface acts
Fig. 2. A collisional contact between triangles. The normal and tangential vectors to the contact line are \( \bar{n} \) and \( \bar{t} \), respectively. The contact line is represented by a dashed line.

In the direction of the relative displacement parallel to the contact line, since the initiation of contact and is proportional to this displacement with coefficient \( K_n L_p \) (2D case), where \( L_p \) is the characteristic size of the elements. Tangential forces can only be withstood up to a frictional limit \( \mu F_n \), where \( \mu \) is friction coefficient. If the tangential force exceeds this limit, the magnitude of this force clips to \( \mu F_n \) but it continues to act in the same direction. Both the tangential and normal forces generated at collisional contacts act through the center of mass of the overlapping area.

This technique is derived from what is often referred to as the “Discrete” or “Distinct” Element Method (but always abbreviated as DEM). However, that name is inappropriate in this context as, within the assembled body, the glued elements are neither discrete nor distinct.

Several element shapes (equilateral triangles, Delaunay triangles, Voronoi polygons) were considered in Refs. 1–3. It was found that, with any of these shapes, the composite material have predictable elastic properties, Young’s modulus \( E \) and Poisson’s ratio \( \nu \), provided that size of the element \( L_p \) is small compared to the size of the body \( L \). The same formulas for \( E^j \) and \( \nu^j \) govern both Delaunay triangles and equilateral triangles and are given (for plane-stress loading) by:

\[
E^j = \frac{4K_n L_p \cdot (K_i / K_n)}{\sqrt{3}(3K_i / K_n + 1)}
\] (1)
\[ \nu^i = \frac{1 - K_i/K_n}{3K_i/K_n + 1} \]  

(2)

where \( L_p \) is the length of a side of the equilateral triangles or the average length of the side of Delaunay triangles. (Here the superscript \(^i\) denotes the elastic properties induced by compliant joints.)

In Refs. 3 and 6, we have used this model to describe breakage of solids by both compression and impact. However, several problems were apparent. First of all, it has trouble modeling large deformations because the rigid elements may not conform well to an arbitrarily deformed shape. Second, there was no way to realistically introduce plasticity into the model. (However, the model will accurately simulate brittle materials for which only limited deformation is possible before the solid fractures.) To overcome these disadvantages, an extension to this model was developed by incorporating portions of the finite element technique to permit the elements to deform their shapes, either elastically or plastically. That extension is described and used in the following.

2. Description of the Model

In the new model, the elements may deform in accordance with a specified elastic or elastic-plastic stress-strain law. So far, only a two-dimensional version of this model has been developed although a three-dimensional equivalent is conceptually possible. At the start, the two-dimensional body will be decomposed into Delaunay triangles. (Triangles are the optimum shape for this type of model as six joints issue from each vertex giving six directions along which a crack might propagate; a higher-order polygon would further restrict the possible directions of crack propagation.) Each triangle is regarded as a finite element with three nodal points situated in the vertices of the triangle. The usual dynamic finite element equation calculates the advance of the nodal points:

\[ \mathcal{M} \frac{d^2 \bar{F}}{dt^2} = F_{\text{ext}} - F_{\text{int}} \]  

(3)

where \( \mathcal{M} \) is the standard \( 6 \times 6 \) consistent mass matrix for triangular finite elements with three nodal points, \( \bar{F} \) is the \( 6 \times 1 \) vector of the nodal displacements, \( F_{\text{ext}} \) is the \( 6 \times 1 \) vector of external forces applied to the triangle through collisional and glued contacts and \( F_{\text{int}} \) is the \( 6 \times 1 \) vector of nodal forces resulting from the deformation of the elements. Note that, unlike the classical finite element method, we do not assemble the global stiffness and mass matrix and are not solving the equation for the whole system; instead we solve the system of the six equations for each element and let the forces inside the body to be transmitted to neighboring elements across the glued joints. In some sense, this new method is intermediate between the old technique described in Refs. 1–3 and a standard finite element method — this method approaches the standard finite element method in the limit of infinitely rigid joints and approaches our old technique in the limit of infinitely rigid elements.
The first step in applying the proposed method is to recalculate the forces on collisional and glued contacts into the $6 \times 1$ vector $\vec{F}_{\text{ext}}$, and, then, to calculate the vector $\vec{F}_{\text{int}}$, according to the proposed stress-strain law. The calculation of $\vec{F}_{\text{ext}}$ is somewhat complicated for collisional contacts, because the compliant boundaries allow the elements to overlap slightly. Figure 3 is a schematic illustration of this process. It is assumed that the force is applied at the center of this contact region (denoted point $C$ in Fig. 3). Suppose that the point $C$ has coordinates $(x_c, y_c)$ and that the force at $C$ is given by the $2 \times 1$ vector $\vec{F}_c = (F_x, F_y)$. First, the element side that is closest to $C$ is found and the point at which the force is applied is moved to the nearest point along that edge — the point $C_1 = (x_{c1}, y_{c1})$ shown in Fig. 3. By implementing this procedure the total force acting on the element does not change and as the overlaps during collisional contacts are small, only a negligible change is made in the total moment acting on the element. Next the vector $(F_x, F_y)$ is decomposed on the vector of nodal forces for the two nodes (here, $N_1$ and $N_2$) that bound the edge side on which the point $C_1 = (x_{c1}, y_{c1})$ resides. The decomposition should satisfy three conditions: (a) the sum of the resulting nodal forces should be equal to $\vec{F}_c$, (b) the moments from the resulting nodal forces around the center of mass of the triangle should be equal to the moments created around the center of mass of the triangle by the force $\vec{F}_c$ and (3) the rate of work done by the resulting nodal forces should be the same as rate of work done by the $\vec{F}_c$. This results in the following set of vector equations:

$$\vec{F}_{N1} + \vec{F}_{N2} = \vec{F}_c$$

(4)

Fig. 3. A schematic illustrating the calculation of the nodal forces due to a "collisional" contact at the point $C$ with force $\vec{F}_c$. 
\[
[\overrightarrow{R_{c1}} \times \overrightarrow{F_c}] = [\overrightarrow{R_{N1}} \times \overrightarrow{F_{N1}}] + [\overrightarrow{R_{N2}} \times \overrightarrow{F_{N2}}]
\]

\[
\overrightarrow{F_c} \cdot (d\overrightarrow{R_{c1}}/dt) = \overrightarrow{F_{N1}} \cdot (d\overrightarrow{R_{N1}}/dt) + \overrightarrow{F_{N2}} \cdot (d\overrightarrow{R_{N2}}/dt)
\]

where \(\overrightarrow{F_{N1}}\) and \(\overrightarrow{F_{N2}}\) are the contributions from this contact to the nodal forces on points \(N_1\) and \(N_2\) and \(\overrightarrow{R_{c1}}, \overrightarrow{R_{N1}}\) and \(\overrightarrow{R_{N2}}\) are vectors connecting center of mass of the triangle with points \(C_1, N_1\) and \(N_2\) (Fig. 3). Values of \(\overrightarrow{F_{N1}}\) and \(\overrightarrow{F_{N2}}\) satisfying the Eqs. (4)–(6) are:

\[
\overrightarrow{F_{N1}} = (1 - L_{c1N1}/L_{N1N2}) \cdot \overrightarrow{F_c}
\]

\[
\overrightarrow{F_{N2}} = (1 - L_{c1N2}/L_{N1N2}) \cdot \overrightarrow{F_c}
\]

where \(L_{N1N2}\) is the distance between points \(N_1\) and \(N_2\), \(L_{c1N1}\) is the distance between points \(C_1\) and \(N_1\) and \(L_{c1N2}\) is the distance between points \(C_1\) and \(N_2\).

Now consider the nodal forces induced by the "glued" contacts. In many ways, this is similar to the procedure for collisional contacts, but is slightly different to accommodate the different way in which this type of contact is handled. Consider an element, such as that shown schematically in Fig. 4, which has a glued joint along the side \(N_1N_2\). Here, the point \(C\) denotes the center of the joint; this may not coincide with the center of the side \(N_1N_2\) as the joint may be partially broken. Next, the total force and moment on this side of the element are separated into the force \(\overrightarrow{F_{c1}}\) and moment around the element’s center of mass \(\overrightarrow{M_{c1}}\) applied to the element by the left half of the joint and the force \(\overrightarrow{F_{c2}}\) and moment \(\overrightarrow{M_{c2}}\) applied to the element by the right half of the joint. Solving the equations:

\[
[\overrightarrow{R_{c1}} \times \overrightarrow{F_{c1}}] = \overrightarrow{M_{c1}}
\]

\[
[\overrightarrow{R_{c2}} \times \overrightarrow{F_{c2}}] = \overrightarrow{M_{c2}}
\]

yields the vectors \(\overrightarrow{R_{c1}}\) and \(\overrightarrow{R_{c2}}\) which point from the element’s center of mass to points \(C_1\) and \(C_2\) on the element edge, \(N_1N_2\), such that forces \(\overrightarrow{F_{c1}}\) and \(\overrightarrow{F_{c2}}\) applied at these points will yield the moments \(\overrightarrow{M_{c1}}\) and \(\overrightarrow{M_{c2}}\) around center of mass of the triangle. Following the procedure described above for collisional contacts the values of \(\overrightarrow{F_{c1}}\) and \(\overrightarrow{F_{c2}}\) are recalculated into contributions to the nodal forces \(\overrightarrow{F_{N1}}\) and \(\overrightarrow{F_{N2}}\) on the triangle.

Next, the internal nodal forces, \(\overrightarrow{F_{int}}\) are determined by applying an appropriate stress-strain law to the element deformations. In a purely elastic case, an isotropic incremental Hooke’s law with constant \(E\) and \(\nu\) is employed:

\[
d\varepsilon_{ik}^e = [(1 + \nu)d\sigma_{ik} - \nu d\sigma_{kk} \delta_{ik}]/E
\]

where \(d\varepsilon_{ik}^e\) is the elastic strain increment in the local coordinate system moving with element and \(d\sigma_{ik}\) is the true (Cauchy) stress increment of in system. (This particular stress-strain law is presented as an example; any stress-strain law could
Fig. 4. A schematic illustrating the calculation of the nodal forces due to a “glued” contact along the side $N_1N_2$.

have been used.) For plastic cases Prandtl–Reiss flow theory is employed so that the total strain increment $d\varepsilon_{ik}$ is sum of the elastic strain increment $d\varepsilon_{ik}^e$ and the plastic strain increment $d\varepsilon_{ik}^p$:

$$d\varepsilon_{ik} = d\varepsilon_{ik}^e + d\varepsilon_{ik}^p$$

(12)

where $d\varepsilon_{ik}^e$ is given by the Eq. (2.9) and $d\varepsilon_{ik}^p$ is proportional to the stress deviator, $\tau_{ik}$:

$$d\varepsilon_{ik}^p = d\lambda \cdot \tau_{ik}$$

(13)

where:

$$\tau_{ik} = \sigma_{ik} - p\delta_{ik}$$

(14)

and:

$$p = -\sigma_{ll}/3.$$  

(15)

The plastic flow is assumed to be incompressible:

$$d\varepsilon_{ll}^p = 0.$$

(16)

As a plastic yield condition we have chosen the modified Coulomb's law:

$$\sqrt{I_2} \leq \text{Coh} + f \cdot p$$

(17)
where $\text{Coh}$ is cohesion, $f$ is friction coefficient of the simulated material and $I_2$ is second invariant of the stress deviator:

$$I_2 = \frac{(\tau_{ik} \tau_{ik})}{2}. \quad (18)$$

As the elements may experience large strains and displacements, the common small-strain/small-displacement finite element will not yield accurate results. The most popular way of dealing with large displacements and strains is to employ an additional geometric stiffness matrix. This was attempted for the current simulation, but was found to be unsatisfactory since it is common that impact induced fracture often produces rapidly rotating fragments consisting of a small number of elements. These rotations lead to numerical errors in the calculation of $\overline{F}_{\text{int}}$ and, most importantly, induce fictitious moments applied to elements by $\overline{F}_{\text{int}}$ that lead to large angular accelerations (as for three-point triangular elements, the internal stresses and strains are constant, $\overline{F}_{\text{int}}$ should not create moments on the elementary triangles). It may be possible to eliminate these moments at each time step, but an easier method presents itself for the triangular elements that are preferred for these simulations.

The technique for calculating $\overline{F}_{\text{int}}$ is based on the concept of natural stresses and strains proposed in Ref. 7 and is applicable to elements (such as the triangles used here) with constant internal stresses and strains. This technique is based on the fact that at any point in two dimensions, three elongations along non-parallel lines completely describe the strain state in the system. Likewise, any three forces acting along non-parallel lines completely describe the stress state of the system. In the case of the constant-stress-constant-strain triangle, it is very convenient to use three elongations of the sides of the triangle divided by the length of the corresponding side as a measure of the triangle’s strain; i.e., referring to Fig. 5, the “natural strain” is given by $\overline{\epsilon}_n = (\Delta L_1/L_1, \Delta L_2/L_2, \Delta L_3/L_3)$, where $\Delta L_i$ represents the extension of triangle’s side $L_i$. In the same way, the “natural stress” vector $\overline{\sigma}_n$ can be composed of the three components of stress that act parallel to the sides of triangle: from Fig. 5, $\overline{\sigma}_n = (\sigma_1, \sigma_2, \sigma_3)$. The advantages of this technique is that as the elongation of the sides of triangle are invariant under a rigid body motion, they will not contribute to the natural strains. Since natural stresses are only calculated using natural strains, no fictitious elastic moments will be created. Furthermore, it is a simple matter to transform these natural stresses of the elements into any Cartesian coordinate system.

Using the results of Ref. 7, it is easy to show that the stress-strain relationship (11) for two-dimensional natural stresses and strains becomes

$$A_n d\sigma_n = E^* d\overline{\sigma}_n \quad (19)$$

where $d\overline{\sigma}_n = (d\sigma_1, d\sigma_2, d\sigma_3)$ is an increment of the $3 \times 1$ vector of natural stresses, $d\overline{\epsilon}_n = (d\epsilon_1, d\epsilon_2, d\epsilon_3)$ or $(dL_1/L_1, dL_2/L_2, dL_3/L_3)$ is an increment of the $3 \times 1$ vector of natural strains, and $E^*$ is the predetermined Young’s modulus of the
elements (which, due to the compliance of the joints, may be different from the Young’s modulus of the composite body). Here, $A_n$ is the $3 \times 3$ stiffness matrix for the $n$th element, which depends somewhat on how the actual three-dimensional stress field is interpreted in two dimensions. Under plane-stress conditions,

$$
A_n = \begin{pmatrix}
1 & (1 + \nu^e)\cos^2 \beta - \nu^e & (1 + \nu^e)\cos^2 \alpha - \nu^e \\
(1 + \nu^e)\cos^2 \beta - \nu^e & 1 & (1 + \nu^e)\cos^2 \gamma - \nu^e \\
(1 + \nu^e)\cos^2 \alpha - \nu^e & (1 + \nu^e)\cos^2 \gamma - \nu^e & 1
\end{pmatrix}
$$

(20)

and under plane-strain conditions,

$$
A_n = \begin{pmatrix}
1 - (\nu^e)^2 & (1 + \nu^e)(\cos^2 \beta - \nu^e) & (1 + \nu^e)(\cos^2 \alpha - \nu^e) \\
(1 + \nu^e)(\cos^2 \beta - \nu^e) & 1 - (\nu^e)^2 & (1 + \nu^e)(\cos^2 \gamma - \nu^e) \\
(1 + \nu^e)(\cos^2 \alpha - \nu^e) & (1 + \nu^e)(\cos^2 \gamma - \nu^e) & 1 - (\nu^e)^2
\end{pmatrix}
$$

(21)

where $\alpha$, $\beta$ and $\gamma$ are the internal angles of the triangle as shown in Fig. 5. This procedure allows calculation of the natural stresses $\sigma_n$ given the increments of the natural strains $\varepsilon_n$. The calculation of the forces $F_n = (F_1, F_2, F_3)$ parallel to the sides of triangle is given by:

$$
F_n = \text{Area} \cdot L_n^{-1} \varepsilon_n
$$

(22)

where $\text{Area}$ refers to the element area and $L_n^{-1}$ is the diagonal matrix:
\( \mathcal{L}^{-1} = \begin{pmatrix} 1/L_1 & 0 & 0 \\ 0 & 1/L_2 & 0 \\ 0 & 0 & 1/L_3 \end{pmatrix} \) \hfill (23)

It is now easy to recalculate the \( 3 \times 1 \) natural force vector \( \overline{F}_n \) to the Cartesian \( 6 \times 1 \) vector \( \overline{F}_{\text{int}} \). Note that this procedure automatically excludes the creation of any elastic moments on the triangular elements.

Now consider the elastic-plastic contributions to the nodal forces \( \overline{F}_{\text{int}} \) which are calculated using the methods proposed in Ref. 8. Suppose the values of forces \( \overline{F}_{\text{int}} \) and \( \overline{F}_{\text{ext}} \) are known, then the displacements during the next time step \( \text{d}t \) can be found by solving the system (3). The next step is to calculate the total Cartesian stresses \( \sigma_{ik}^\text{el} \) (relative to a fixed coordinate system) inside each element assuming that the deformation is completely elastic. These stresses are then checked, element by element, against the the criterion for plastic yielding (17). If satisfied, the elements behave elastically and there is nothing left to be done. If the plastic limit is exceeded, the new Cartesian stresses inside each triangle are calculated using the formulae:

\[
\sigma_{ik} = \tau_{ik} + p
\]

\[
\tau_{ik} = \tau_{ik}^\text{el} \cdot (\text{Coh} + f \cdot p)/\sqrt{I_2^\text{el}}
\] \hfill (25)

where the superscript "el" corresponds to the values calculated assuming perfectly elastic behavior and \( p \) is minus one-third the first stress invariant (15). Note that from the incompressibility condition (16), \( \epsilon_{il} = \epsilon_{il}^\text{el} \), and that the value of \( p \) can be calculated independently of the values \( \sigma_{ik} \) simply by calculating the change of the element area with respect to its initial area. After this the invariant stresses \( \overline{\sigma}_n \) are calculated and the procedure continues.

The procedure described here has only first order of accuracy with respect to time, but as will be discussed in Sec. 3 of this paper, the time step must be kept small for other reasons, so that a more accurate technique becomes unnecessary.

### 3. The Elastic Properties of the Composite Body

As mentioned above, the elastic properties of the composite body will depend on both the elastic properties, \( E^e \) and \( \nu^e \), of the elements and on the elastic properties, \( E^j \) and \( \nu^j \), implied by the compliance of the joints. Consider a small deformation \( \epsilon_{ik} \), of a body consisting of a large number of deformable Delaunay triangles. Thus, the portion of the global deformation caused by deformation of the elements \( \epsilon_{ik}^e \) and the portion caused by the deformation of the joints \( \epsilon_{ik}^j \) are also small. The global deformation must be a sum of the two contributions:

\[
\epsilon_{ik} = \epsilon_{ik}^e + \epsilon_{ik}^j.
\] \hfill (26)

If the body is in equilibrium, the stresses created by the joints and stresses inside particles should be identical and can be denoted by a single stress tensor, \( \sigma_{ik} \). Since
small deformations are assumed, the shapes of the elements do not radically change so that the moduli arising from the compliant joints, $E^j$ and $\nu^j$, are still described by Eqs. (1)–(2). Using the Hooke’s law for both the elements and joints:

$$\epsilon_{ik} = \epsilon^e_{ik} + \epsilon^j_{ik} = \frac{[(1 + \nu^e)\sigma_{ik} - \nu^e \sigma_{ll} \delta_{ik}]}{E^e} + \frac{[(1 + \nu^j)\sigma_{ik} - \nu^j \sigma_{ll} \delta_{ik}]}{E^j}. \quad (27)$$

Rearranging Eq. (27) yields:

$$\epsilon_{ik} = \frac{[(1 + \nu)\sigma_{ik} - \nu \sigma_{ll} \delta_{ik}]}{E} \quad (28)$$

where

$$E = \frac{E^e E^j}{E^e + E^j} \quad (29)$$

and

$$\nu = \frac{\nu^e E^j + \nu^j E^e}{E^e + E^j} \quad (30)$$

which is the equivalent of Hooke’s law for the composite material. It is easy to see from Eqs. (29) and (30) that $E$ and $\nu$ approach $E^j$ and $\nu^j$ as the ratio $E^j/E^e$ goes to zero (particles becomes rigid), and that $E$ and $\nu$ approach $E^e$ and $\nu^e$ as the ratio $E^j/E^e$ goes to infinity (joints becomes rigid), exactly as might be expected. This expectation is confirmed numerically in Fig. 6 (Poisson’s ratio) and Fig. 7 (Young’s modulus) for a square sample divided into 248 Delaunay triangles. The results shown correspond to $K_n/K_t = 1 \ (\nu^j = 0), \ \nu^e = 0.3$, assuming plane stress.

[Image of a graph showing the modulus of elasticity, $E/E^e$, against $(\epsilon/E^e)$ with various data points and a dashed line indicating theoretical predictions.]

Fig. 6. Theoretical (solid line) and simulated (markers) Young’s moduli.
conditions. These results are plotted as \( E/E^c \) and \( \nu/\nu^c \) and may be found by simple manipulations of Eqs. (29) and (30):

\[
\frac{E}{E^c} = \frac{E^j/E^c}{1 + E^j/Ee}
\]

\[
\frac{\nu^j}{\nu^c} = \frac{E^j/E^c}{1 + E^j/E^c}
\]

and can be seen to be in close agreement with the simulated results.

In many ways, it is preferable that the elastic properties of the composite be determined by the element properties, \( E^c \) and \( \nu^c \). In particular, this dictates that the elastic properties will not change even if breakage occurs down to the level of single elements. However, \( E \) and \( \nu \) converge to \( E^c \) and \( \nu^c \) in the limit of large \( E^j/E^c \); for \( E^j/E^c = 10 \) the difference is about 9% and for \( E^j/E^c = 100 \) the difference is less than 1%. However, large \( E^j \) requires a small integration step size, \( dt \), for stability of the numerical integration. This time step is much smaller than is required by the elastic properties of the individual elements; consequently, the first order technique used to calculate the internal forces, (described in Sec. 2) cannot lead to large numerical errors.

4. The Work of Fracture

The new model proposed in this paper has a lot of advantages compared to the rigid element model described in Refs. 1–3, but it also has one problem which is related
to the energy consumed during breakage. By thermodynamic reasoning, Griffith showed that the decrease of the elastic energy caused by crack propagation should be larger or equal to the "work of fracture" (or the energy absorbed in the creation and growth of a crack), which in most cases can be assumed to be a material property. It is generally accepted that, while the propagation of the crack is almost completely described by energy considerations, the initiation of the breakage depends on the stress concentration developed around internal microscopic flaws which globally determine the tensile strength of material, (which is also typically assumed to be a material constant). So to properly describe the initiation and propagation of the crack in our model, two parameters are required, the tensile strength, \( \sigma_{\text{tens}} \), and the work of fracture, \( W_{\text{cr}} \).

The problem, which also existed to a lesser extent in the rigid element model, is that the only lost energy is that stored in the compliant joint. If any portion of a joint is broken as soon as \( \sigma > \sigma_{\text{tens}} \), then \( \sigma_{\text{tens}} \) and \( W_{\text{cr}} \) are not independent, but are related by:

\[
W_{\text{cr}} = \frac{\sigma_{\text{tens}}^2}{2K_n}
\]  

(33)

This was not a major concern for rigid elements, as all of the elastic energy of the body is stored inside the joints and is released as the joint breaks. Thus a significant portion of the elastic energy is released as the crack propagates. Unfortunately the same cannot be said of the new model, as much of the elastic energy will be stored within the elastic elements themselves. As stated earlier, very stiff glued joints (i.e., large \( K_n \)) are used so that the global elastic properties are largely determined by the element properties; thus, if \( W_{\text{cr}} \) is to have a realistic value, \( \sigma_{\text{tens}} \) must be unrealistically large and if \( \sigma_{\text{tens}} \) has a realistic value, \( W_{\text{cr}} \) will be unrealistically small. The resolution of this conflict involves developing a scheme to release some of the energy stored in the elements as the crack propagates, so that the energy loss is not entirely dependent on the joint properties.

The essence of the idea is to limit the speed of the crack propagation, \( V_{\text{cr}} \), along the joint. The initiation of fracture will still occur at \( \sigma_{\text{tens}} \), but, near the end of breakage, the joint will be over extended, storing more energy which will be lost when the joint breaks. From one point of view, this also fixes a physical inaccuracy implicit in this simulation technique. The use of triangular elements with nodes only at their vertices implies that the stresses inside the elements are constant. Thus, the tensile stress on the joints must also nearly be constant, so that the tensile strength will be exceeded at nearly every point along the joint at almost the same moment. This may lead to unrealistically large crack propagation rates, which, physically, should be limited by the elastic wavespeed of the surface waves. (This would not necessarily be the case for the rigid element model for which, as is illustrated in Fig. 1, part of the joint may be in tension while another part may be in compression.)
Fig. 8. Schematic of the situation used to estimate the energy released during breakage of a glued joint.

It is possible to approximate the energy release in breakage by using a very simple model which is shown schematically in Fig. 8. Assume that two adjacent elements may be represented by two semi-infinite elastic bodies with vertical size $L_p$ and elastic properties $E$ and $\nu$. Assume furthermore, that the two bodies are connected together by a glued joint with stiffness $K_n$. At $t = 0$, the joint spans the length, $L_p$, of the joint and the horizontal stress inside the joint is tensile and equal to $\sigma_{\text{tens}}$ while the vertical stress is zero. Subsequently, the joint begins to break at a constant speed $V_{cr}$, so that, at time $t$, the length of the remaining unbroken joint is

$$l = L_p - V_{cr}t.$$  \tag{34}

It is assumed that the boundaries of the bodies connected by the joint do not deform or rotate during the process. This will somewhat limit the validity of the calculation as a partially broken joint will induce a moment that, given enough time, will rotate the elements; thus, accurate results will only be obtained for rapid breakage.

The progressive elimination of the joint creates elastic waves moving outward from the boundaries through the semi-infinite bodies. Assuming no deformation of either the vertical or horizontal boundaries, these waves will be simple plane waves moving in the $x$-direction. In this case the equation of motion is:

$$\rho \frac{\partial v}{\partial t} = \frac{\partial \sigma}{\partial x},$$  \tag{35}

where $x$ is the horizontal coordinate, $\sigma$ is the horizontal stress, $\rho$ is the density of the bodies and $v$ is the horizontal material velocity induced by the wave. Consider the wave passing through the boundary of the semi-infinite body that lies to the left of the joint in Fig. 8. For a wave moving into the body, away from the boundary, $\partial v/\partial t = C \partial \sigma/\partial t$, where $C = \sqrt{E/\rho}\left(1 - \nu^2\right)$ is speed of a longitudinal elastic wave.
The corresponding equation of motion becomes:

$$\rho C \frac{\partial v}{\partial x} = \frac{\partial \sigma}{\partial x}$$  \hspace{1cm} (36)$$

which may be integrated to yield

$$\sigma = \text{const} + \rho C v.$$  \hspace{1cm} (37)$$

This may be used to determine the stress on the vertical boundary of the left body. If $y$ is the “elongation” of the joint, (i.e., the distance between the two semi-infinite bodies), the velocity at which the joint is stretching is $dy/dt$; if both bodies move with the same speed, the velocity of the surface of the left body is $v = -0.5(dy/dt)$. Furthermore, the initial stress on the surface must be the tensile strength, $\sigma_{\text{tens}}$. Thus, during breakage, the stress on the boundary is $\sigma_{\text{tens}} = \rho C(dy/dt)/2$ which must equal the stress supported by the joint $K_n ly/L_p$, i.e.:

$$\sigma_{\text{tens}} - \rho C \frac{dy}{2} \frac{dt}{dt} = K_n \frac{ly}{L_p}.$$  \hspace{1cm} (38)$$

As the crack propagates with velocity $V_{cr}$, $dy/dt = -V_{cr}(dy/dl)$

$$\sigma_{\text{tens}} + \rho C V_{cr} \frac{dy}{2} \frac{dl}{dl} = K_n \frac{ly}{L_p}.$$  \hspace{1cm} (39)$$

which is an ordinary first-order differential equation. The solution is:

$$y(l) = \exp\left[\frac{K_n l^2}{\rho CV_{cr} L_p}\right] \cdot \left[-\frac{2\sigma_{\text{tens}}}{\rho CV_{cr}} \int_{0}^{l} \exp\left[-\frac{K_n x^2}{\rho CV_{cr} L_p}\right] dx + \text{const}\right].$$  \hspace{1cm} (40)$$

Using the initial condition $y(L_p) = \sigma_{\text{tens}}/K_n$:

$$y(l) = \frac{\sigma_{\text{tens}}}{K_n} \left[\exp[\phi^2(1 - (l/L_p)^2)] + 2\phi \cdot \exp[(\phi l/L_p)^2] \int_{\phi l/L_p}^{\phi} \exp(-x^2)dx\right]$$  \hspace{1cm} (41)$$

where:

$$\phi = \sqrt{K_n L_p/(\rho CV_{cr})}.$$  \hspace{1cm} (42)$$

The value of $y(l)$ for any given $l$ can be easily calculated numerically from Eq. (41). Once $y(l)$ is known, it is possible to calculate the total energy, $\mathcal{E}_{cr}$, released during the breakage of the joint:

$$\mathcal{E}_{cr} = (K_n/2) \int_{0}^{L_p} y^2(l) dl = (K_n L_p/2) \int_{0}^{1} y^2(l/L_p) d(l/L_p)$$  \hspace{1cm} (43)$$

or, rewriting in terms of the energy release per unit length $W_{cr} = \mathcal{E}_{cr}/L_p$,

$$W_{cr} = (K_n/2) \int_{0}^{1} y^2(l/L_p) d(l/L_p).$$  \hspace{1cm} (44)$$
Equation (41) can be rewritten as

\[ y(l) = (\sigma_{\text{tens}}/K_n) \cdot F(\phi, l/L_p) \]  

(45)

where \( F \) is a dimensionless function of the two dimensionless arguments \( l/L_p \) and \( \phi \) given by:

\[ F(\phi, l/L_p) = \exp(-\phi^2(1 - (l/L_p)^2)) + 2\phi \cdot \exp((\phi l/L_p)^2) \int_{\phi l/L_p}^{\phi} \exp(-x^2)dx \]  

(46)

So Eq. (44) becomes:

\[ W_{cr} = \frac{\sigma_{\text{tens}}^2}{2K_n} \int_0^1 F^2(\phi, l/L_p)d(l/L_p) \]  

(47)

Equation (47) shows that the dimensionless work of fracture \( W_{cr}^d = W_{cr}/(\sigma_{\text{tens}}^2/(2K_n)) \) depends only on value of \( \phi \). Numerically calculated values of \( W_{cr}^d(\phi) \) are presented in Fig. 9 by the solid line. It is possible to see from this figure that for the values \( 2 < \phi < 25 \), \( W_{cr}^d(\phi) \) is very close to the straight line:

\[ W_{cr}^d = 2.251\phi - 1.197 \]  

(48)

(indicated by a dashed line in Fig. 9), which makes numerical calculation of \( W_{cr}^d \) unnecessary for all but very small \( \phi \). Indeed most physically reasonable parameters will fall into the linear range; i.e., \( E'/E'' = 100 \), \( \nu = 0.3 \), and \( V_{cr} = 0.9C_t \), corresponds to \( \phi = 17.18 \) (and, incidentally, corresponds to \( W_{cr}^d = 37.42 \)). Here, \( C_t \) is speed of the transversal elastic waves:

\[ C_t = \sqrt{\frac{E}{2\rho(1 + \nu)}} \]  

(49)

which is slightly larger than the physical limit to the crack propagation speed. The value \( W_{cr}^d = 37.42 \) means that the energy release during the propagation of the crack through the body for these parameters is 37.42 times larger than when the joint breaks instantaneously along any portion where the normal stress exceeds the tensile strength.

Note that Eq. (4.15) is actually a rather approximate estimate for the energy released during the breakage of the joint between two of the kinds of elements that are used in the simulation. First, as mentioned above, it is supposed that the elements do not rotate, despite the fact that a partially broken joint, by exerting a tensile force only over its unbroken part, will apply a moment to the elements. In fact as rotation stretches the fibers at the tip of the crack, it may be a principal driver of the crack propagation. However, this effect will be minimized if the crack propagates so quickly that the elements can exhibit no appreciable rotation. The
other source of error in the analysis arises from the assumption of infinite element sizes. In the derivation of the Eq. (41) it is assumed that media around the joint is perfectly elastic. But since the crack speed will be of the same order as the sound speed, the size of the elastic wave generated during this breakage will be roughly the size of an element and cannot be accurately reproduced within the framework of a finite element technique.

Several simulations were performed to investigate the importance of these two effects and to check the accuracy of Eq. (4.15). The simulations were performed on

![Graph showing dimensionless work of fracture as a function of \( \phi \).](image)

**Fig. 9.** Theoretical (solid line) and simulated (markers) values of the dimensionless work of fracture \( W_{d}^{*} = W_{d} / (\sigma_{tens}^{2} / (2K_{t})) \) as a function of \( \phi = \sqrt{K_{t}L_{p}} / (\rho C_{Vc}) \). Note that, for a given value of \( V_{cr}/C_{t} \) the work of fracture lies on straight lines.

![Diagram of a rectangular body into 1278 elements used to gather the simulated work of fracture data shown in Fig. 9.](image)

**Fig. 10.** The decomposition of a rectangular body into 1278 elements used to gather the simulated work of fracture data shown in Fig. 9.
square samples, with side length $L$, that are composed of 1278, 2238 and 3790 Delaunay triangles (see Fig. 10 for a diagram of the body divided into 1278 triangles). To ensure the location and length of the crack produced, all of the joints are made unbreakable except along a vertical line of joints which is purposely placed through the center of the body. The vertical faces of the sample are glued to rectangular blocks which are slowly moved apart to create quasistatic tension within the sample. Plane-stress conditions are assumed. The theoretical and computational results are compared in the Table 1 and in Fig. 9. It is possible to see from the Table 1 that the dimensionless work of fracture obtained from the simulations is nearly independent of the value of Poisson’s ratio, the ratio of the tensile strength to the Young’s modulus $\sigma_{\text{tens}}/E$, and there is no consistent trend with the size of the element $L_p$ (provided that this size is small compared to the body size $L$, and that $K_n L_p$ is

<table>
<thead>
<tr>
<th>$E^*/E^\ell$</th>
<th>$\sigma_{\text{tens}}/E^*$</th>
<th>$V_{\text{cr}}/C_t$</th>
<th>$\nu_e$</th>
<th>$(W_{\text{cr}}^c)_d$</th>
<th>$L_p/L = 4.39 \cdot 10^{-2}$</th>
<th>$L_p/L = 3.32 \cdot 10^{-2}$</th>
<th>$L_p/L = 2.55 \cdot 10^{-2}$</th>
</tr>
</thead>
<tbody>
<tr>
<td>$E^*/E^\ell = 0.01$</td>
<td>$\sigma_{\text{tens}}/E^* = 5 \cdot 10^{-4}$</td>
<td>$V_{\text{cr}}/C_t = 0.9$, $\nu_e = 0.3$</td>
<td>37.48</td>
<td>41.53</td>
<td>+9.7%</td>
<td>36.23</td>
<td>-3.4%</td>
</tr>
<tr>
<td>$E^*/E^\ell = 0.01$</td>
<td>$\sigma_{\text{tens}}/E^* = 5 \cdot 10^{-4}$</td>
<td>$V_{\text{cr}}/C_t = 0.45$, $\nu_e = 0.3$</td>
<td>53.70</td>
<td>87.37</td>
<td>+38.5%</td>
<td>86.65</td>
<td>-38.0%</td>
</tr>
<tr>
<td>$E^*/E^\ell = 0.01$</td>
<td>$\sigma_{\text{tens}}/E^* = 5 \cdot 10^{-4}$</td>
<td>$V_{\text{cr}}/C_t = 1.35$, $\nu_e = 0.3$</td>
<td>30.35</td>
<td>25.89</td>
<td>-17.2%</td>
<td>25.87</td>
<td>-17.2%</td>
</tr>
<tr>
<td>$E^*/E^\ell = 0.01$</td>
<td>$\sigma_{\text{tens}}/E^* = 3 \cdot 10^{-4}$</td>
<td>$V_{\text{cr}}/C_t = 0.9$, $\nu_e = 0.3$</td>
<td>37.47</td>
<td>43.99</td>
<td>+2.7%</td>
<td>38.47</td>
<td>+2.6%</td>
</tr>
<tr>
<td>$E^*/E^\ell = 0.01$</td>
<td>$\sigma_{\text{tens}}/E^* = 5 \cdot 10^{-4}$</td>
<td>$V_{\text{cr}}/C_t = 0.9$, $\nu_e = 0.2$</td>
<td>37.22</td>
<td>40.37</td>
<td>+7.8%</td>
<td>35.66</td>
<td>-4.4%</td>
</tr>
<tr>
<td>$E^*/E^\ell = 0.01$</td>
<td>$\sigma_{\text{tens}}/E^* = 5 \cdot 10^{-4}$</td>
<td>$V_{\text{cr}}/C_t = 0.9$, $\nu_e = 0.4$</td>
<td>37.42</td>
<td>41.00</td>
<td>+8.7%</td>
<td>36.45</td>
<td>-2.7%</td>
</tr>
<tr>
<td>$E^*/E^\ell = 0.1$</td>
<td>$\sigma_{\text{tens}}/E^* = 5 \cdot 10^{-4}$</td>
<td>$V_{\text{cr}}/C_t = 0.9$, $\nu_e = 0.3$</td>
<td>11.08</td>
<td>12.31</td>
<td>+10.6%</td>
<td>11.73</td>
<td>+5.5%</td>
</tr>
</tbody>
</table>
held constant so that $E^t$, as given in Eq. (1), does not change). However, the work of fracture seems to be strongly dependent on the ratio of the speed of the crack propagation to the transversal sound speed $V_{cr}/C_t$. From Fig. 9, it can be seen, that the analytical result, Eq. (4.15), only accurately predicts the work of fracture if $V_{cr}/C_t$ is close to one. None the less, the analysis does show the importance of the parameter, $\phi$, and all of the simulated values do appear to vary linearly with $\phi$, as does the theoretical result. By curve fitting, a good approximation is:

$$W_{cr}^d = (1.345(V_{cr}/C_t)^2 - 4.333(V_{cr}/C_t) + 5.364)\phi - 1.197$$  \hspace{1cm} (50)

which is shown as dashed lines in Fig. 9. As a result, by controlling $V_{cr}$, (which affects both $\phi$ and $V_{cr}/C_t$) it is possible to change the work of fracture independently of the tensile strength, $\sigma_{tens}$.

A close examination shows how the effects of element rotation and finite size cause the deviation between the simulated and theoretical results in Fig. 9. This is illustrated in Fig. 11 which shows the dimensionless elongation of the joint $y(l)/(\sigma_{tens}/K_n)$, plotted as a function of the dimensionless broken length, $(L_p - l)/L_p$ (thick solid line) for $\phi = 14.0$. (That is, as $l$ represents the remaining unbroken length of the joint, $(L_p - l)/L_p = 0$ corresponds to a completely intact joint while $(L_p - l)/L_p = 1$ corresponds to a completely broken joint.) Also plotted are the simulated values taken from an element near the center of the body from a simulation using 1278 Delaunay triangles and $V_{cr} = 0.9C_t$. As the analysis assumes.

![Fig. 11. Theoretical value of the dimensionless joint elongation $y(l)/(\sigma_{tens}/K_n)$ vs $l/L_p$ for $\phi = 14.0$ (thick solid line) and computational values of the elongations of the center of the joint (thin solid line) and both ends of the joint (dashed and dotted lines) for the case of $V_{cr} = 0.9C_t$, 1278 Delaunay triangles. This illustrates the two sources of error.](image-url)
that the joint does not rotate, the elongation is constant along the joint. However, due to the rotation of the element, the elongation will be greatest at the end of the joint from which the crack propagates, (dotted line) and smallest at the opposite end (dashed line); the distance between these two lines is representative of the rotation of the element. The elongation at the center of the joint is represented by the thin solid line. (The breakage changes the actual locations of the ends of the glued section of the joint. As Fig. 11 shows that the joint has zero length at the end of breakage, so that the center of the joint and the two ends converge to a single physical point and, of course, to a single elongation.) Note that the elongation is nonzero even when the joint is unbroken, \((L_p - l)/L_p = 0\); this is due to the tensile force applied to the joint that initiates the fracture. For the same reason, the joint becomes extremely elongated near the end of breakage as the tensile force, originally supported by the entire side, is now supported by only a small portion of the joint. Figure 11 shows that the theoretical curve increases very rapidly up to the value of about 25 as \((L_p - l)/L_p \to 1\). This rapid rise is only vaguely followed by the simulated points. But note that the rise occurs only in approximately the last 20% of the joint breakage. As the joint breaks at nearly the sound speed, the rise is reflected in elastic waves that could only have progressed about 20% of an element away from the joint before the joint is completely broken. Waves of sub-element size cannot of course be accurately represented in the simulation and can be seen to result in much smaller joint elongation and thus smaller work of fracture. Obviously, the larger the crack speed, the smaller the wave that is generated and the smaller the simulated work of fracture; this can be seen in Fig. 9 as the simulated data for \(V_{cr}/C_t\) greater than one, lies below the theoretical line. The effect of the particle rotation can be seen by comparing the work done by the different elongations of the two ends of the joint. As a result of this rotation, the elongation of the end of the joint where the fracture initiates, is larger than the theoretical prediction, while the elongation of the opposite end of the joint initially drops almost to zero and starts to increase only near the end of breakage. As, for most of the breakage, the part of the joint that is breaking is elongated further than theoretically predicted, more energy is released as that portion of the joint breaks. Consequently, the effect of the rotation leads to more energy loss than predicted by Eq. (47). These two sources of error work against each other, so that Eq. (47) still yields reasonable ballpark values for the fracture energy.

In real solids, the work of fracture is the work required to generate the extensive microscopic surface damage that occurs during rupture. It is convenient, from a continuum point of view, to model this as a plastic phenomenon. However, it may be independent of the macroscopic plasticity that is typically accomplished by the deformation of millimeter scale crystals in a polycrystalline metal. In this way, the current model mimics real behavior better than most continuum models as it separates continuum plasticity (i.e., plastic deformation of the elements) from the non-continuum rupture work performed at the element boundaries during crack propagation.
5. Examples

This simulation technique was developed to study the growth of multitudinous cracks that lead to particle fragmentation in comminution systems. Many of those systems use impact as a means of inducing breakage. Consequently, a normal collision between a circular particle and a rigid plate will be used to demonstrate the simulation in action. In all cases, plane-strain is assumed and $E'/E^p = 100.0$, $\nu = 0.3$, and $K_n/K_t = 1$. The particle has been divided into 1868 Delaunay triangles so that $L_0/L = 3.21 \cdot 10^{-2}$. The friction coefficient is taken to be $\mu = 0.3$. The ratio between the impact velocity, $V_o$, and the sound speed, $C$, is taken to be large, $V_o/C = 0.136$. It is shown that $V_o/C$ represents a characteristic particle deformation; here, this relatively high initial velocity of the particle has been chosen to force a large particle deformation and, thus, to demonstrate the particular advantages of the present model with respect to the rigid particle model.

First, consider a perfectly elastic collision between the particle and plate. The tensile strength and plastic cohesion, Coh, have been chosen to be large enough to avoid both breakage and plasticity. This is studied to demonstrate that the composite particle behaves as would be expected and to explain the breakage observed in the subsequent simulation shown in Fig. 14. The results of this collision are shown on Fig. 12(a) (particle deformation history), Fig. 12(b) (principal tensile stress history) and Fig. 12(c) (principal compressive stress history). One line is plotted for each element in the stress figures; the direction of the line represents the direction of the principal stress and the length represents its magnitude. The ability to resolve these stresses on the element level is an added advantage over the rigid element model. (Note that the first panel in Fig. 12(a) has no corresponding panel in Fig. 12(b) and Fig. 12(c), since the stresses are zero at the initiation of impact and it would be wasteful to demonstrate something so obvious. But, to take maximum advantage of the available space, the fourth panels in Fig. 12(b) and Fig. 12(c) occur at a later time than any panel in Fig. 12(a).) At the start of the collision, the particle flattens upon contact with plate transferring its kinetic energy to the potential energy of elastic deformation. At this point, large horizontal tensile stresses can be observed near the center of the particle. Note also, that there is a region immediately surrounding the contact point where the friction between the particle and the plate freezes out the tensile stresses. In the next frame, the elastic energy transforms back to the kinetic energy, and the particle departs from the plate. At this moment of time it is extended along the vertical axes, which creates vertical tensile stresses. Once contact has been lost with the plate, the particle begins a long series of oscillations. Due to the energy that remains in these oscillations the restitution coefficient for this collision is 0.876. The accuracy of calculations, checked by calculating the energy balance in the system, is 1.2%.

Next, consider an elastic-plastic collision between a particle and a rigid plate. All parameters of this collision are the same except that the plastic limit is lowered to Coh/$E = 5 \cdot 10^2$. (Again this was performed to show that the simulated particle
behaves as expected and to explain the breakage observed in the subsequent simulation shown in Fig. 15.) The results are shown in Fig. 13(a) (deformation history), Fig. 13(b) (principal tensile stress history) and Fig. 13(c) (principal compressive stress history). The collision creates large plastic deformation about the contact point and reduces the restitution coefficient to a value of about 0.213. Of this, 78% of the initial kinetic energy of the particle dissipates to plasticity; the other 17% of the lost kinetic energy remains in the oscillations of the particle. The plasticity inside the particle also strongly reduces both tensile and compressive stresses compared to the elastic case. The maximum tensile stresses no longer occur at the center of particle, but instead, are observed right next to the particle boundary at the ends of the line of contact; this is extremely difficult to perceive.
in this figure due to the proximity of the stress line to the particle boundary and is, perhaps, most easily seen as a locally thicker boundary line. Note that, due to the large plastic deformation, the particle exhibits only barely noticeable oscillations of its shape after leaving the surface. The accuracy of the energy balance in the system is 0.2%.

Now these two cases will be repeated, but this time the tensile strength between the elements is reduced to allow the particles to break. The first case is the brittle analog to the elastic case shown in Fig. 12; all parameters of this collision are the same except for a reduced tensile strength, $\sigma_{\text{tens}}/E = 0.19$. This value of the tensile strength is very close to the maximum tensile stress observed near the center of the particle during the purely elastic collision so that little overall breakage might be
Fig. 14. Fracture history resulting from a brittle-elastic collision with a rigid plate. This is the same collision shown in Fig. 12, except that the tensile strength has been reduced until it is just slightly larger than the maximum internal tensile stresses observed in that collision.

Fig. 15. Fracture history resulting from a brittle-elastic-plastic collision with a rigid plate. This is the same collision shown in Fig. 13, except that the tensile strength has been reduced until it is just slightly larger than the maximum internal tensile stresses observed in that collision.
anticipated. The fracture follows the large central tensile stresses so that the particle divides into two parts along a vertical crack (see the time history in Fig. 14). Once the particle has lost contact with the plate, it was seen to go through a series of elastic oscillations; these oscillations generate vertical tensile stresses which cause the further development of horizontal cracks.

The fragmentation analog to the plastic collision case presented in Fig. 13 is shown in Fig. 15; all the parameters are the same except that the tensile strength is reduced to $\sigma_{\text{tens}}/E = 0.11$. (The tensile strength here must be smaller than for the elastic case shown in Fig. 14 as the plasticity reduces the magnitude of the induced tensile stresses.) As in the elastic case, the body first divides at the point of maximum tensile stress, which, in this case occur near the edges of the contact line and result in the small cracks that appear there. However, little fracture occurs while the particle is in contact with the plate. Pervasive fracture occurs along roughly horizontal lines as the internal elastic stresses relax.

6. Conclusions

This paper describes a new simulation of solid deformation and fracture. The technique builds upon an earlier rigid element simulation.\textsuperscript{1-3} Here, portions of the finite element technique are employed at the element level to allow the elements themselves to deform. As a result, the simulation is applicable to situations involving large elastic and plastic deformations. The elastic properties were found to be simple combinations of the elastic properties of the elements and of the elastic properties implied by the compliance of the joints. By varying the crack propagation speed, it is possible to change the work of fracture independently of the tensile strength of the joint, in a manner that may be approximated theoretically, as long as the crack propagation speed is kept reasonably close to the sound-speed within the material. Finally, several examples were presented of the impact-induced breakage of round particles. The results show that the pattern of breakage may be explained in terms of the stresses induced within the particles. Additional advantages are obtained if the joint stiffness is made large. In that case, the elastic properties of the composite body are determined by the element properties, and are thus preserved, even when the body has fragmented into individual elements. Also, stiff joints mean that the internal stresses are supported by the deformation of the elements and can be locally determined by the intra-element stresses.

Acknowledgments

This work was supported by a grant from the International Fine Particle Research Institute. Special thanks to Holly Campbell for proofreading the manuscript.

References