Critical Time Step Estimation for Three-Dimensional Explicit Impact Analysis
R. F. Kulak
Reactor Analysis and Safety Division
Argonne National Laboratory
Argonne, Illinois 60439-4842

ABSTRACT

A method to obtain a bound on the critical time step of impact problems that employ explicit integration and penalty type contact elements is presented. The method uses a Gerschgorin bound obtained from a partially assembled stiffness matrix for the nodes that belong to elements involved in the contact region. A numerical example is presented to illustrate the methods effectiveness.

INTRODUCTION

The modeling of impact between reactor components must be taken into account in many reactor safety assessments that involve transient loadings. Finite element analysis provides a tractable approach to solving general nonlinear mechanics problems, and the use of adaptive penalty interface elements provide a means of taking into account the specific nonlinearities that occur during impact between two bodies. For nonlinear transient problems, an explicit temporal integrator (e.g., central difference) is often used to obtain the temporal evolution of the impact problem. A disadvantage of explicit methods is that they are conditionally stable and, thus, required that the computational time step be smaller than the critical value.

For structural and continuum elements a bound on the critical time step can normally be obtained from the highest eigenvalue of the unconstrained elements. However, the penalty elements have zero mass and their resulting element frequencies are infinity. Thus a bound for the critical time step cannot be obtained from the unconstrained frequency for a penalty element.
Because of this ad hoc trial and error methods are often employed which can lead to overly conservative estimates that increase the computational cost. An approach by Plesha [1] uses the power method (e.g. Isaacson and Keller [2]) to determine the maximum eigenvalue for a finite element mesh.

Here, a method to obtain a bound on the critical time step is presented that uses a partially assembled global stiffness matrix assembled from the element stiffness matrices of the penalty elements and their contiguous neighbor elements. The method has been incorporated into the general-purpose, nonlinear, finite element code NEPTUNE [3]. One example is presented to illustrate the method's effectiveness. The example is the axial impact of two bars.

GOVERNING EQUATIONS

The semidiscretized equations of motion for contacting bodies can be obtained by applying the principal of virtual power over the bodies (e.g., Kulak [4]) to give

\[
m_{ij} \ddot{u}_{i} + f_{j}^{\text{int}} = f_{j}^{\text{ext}}
\]

where \(m_{ij}\) is a lumped (i.e. diagonal) nodal mass matrix, \(u_i\) is the displacement, \(f_{j}^{\text{int}}\) is the internal nodal force, which includes contributions from the deformable bodies and the penalty contact elements, and \(f_{j}^{\text{ext}}\) is the external nodal force. The lower case subscripts refer to the global coordinate directions, and the upper case subscripts refer to node numbers.

Equation (1) is explicitly integrated in time using the central difference operator. The acceleration at time \(t\) is computed from equation (1) and the velocities and displacements are time advance according to

\[
\ddot{u}_{i} (t + 0.5 \Delta t) = \ddot{u}_{i} (t - 0.5 \Delta t) + \Delta t \dddot{u}_{i} (t)
\]

\[
u_{i} (t + \Delta t) = u_{i} (t) + \Delta t \dot{u_{i}} (t + 0.5 \Delta t)
\]

The above algorithm is relatively simple. However, it is only conditionally stable, and for undamped linear systems the time step is limited to

\[
\Delta t \leq \Delta t_{\text{crit}} = \frac{2}{\omega_{\text{max}}}
\]
where \( \omega_{\text{max}} \) is the highest frequency of the discretized system.

Since we are interested in estimating a starting value for the time step, the materials are assumed to be linear, and geometric nonlinearities do not have to be considered. For a linear material, the stiffness form of equation (1) is given by

\[
m_{ij} \ddot{u}_i + k_{ij} u_i = f_{ij}^{\text{ext}}
\]  

(4)

where \( k_{ij} \) is the global stiffness matrix.

The associated eigenproblem for equation (4) is

\[
m_{ij} \phi_i - \lambda k_{ij} \phi_i = 0
\]

(5)

where \( \lambda \) is the eigenvalue and \( \phi_i \) is the eigenvector. Note the eigenvalue is the square of the frequency, \( \lambda = \omega^2 \).

**ELEMENT-BASED STABILITY CRITERIA**

For problems that do not involve contact, a conservative bound on the maximum eigenvalue of equation (5) can be obtained from the eigenproblem of the unconstrained element (e.g., Hughes [5]), that is,

\[
\lambda_{e,\text{max}} \leq \max_e \lambda_e
\]

(6)

where \( \lambda_e \) is the eigenvalue for a single unconstrained element; namely

\[
k_{ij}^{e} \phi_i^{e} = \lambda_m m_{ij}^{e} \phi_i^{e}
\]

(7)

Thus, considering only the non-impacting elements, the critical time step is given by

\[
\Delta t_{e,\text{crit}} = \frac{2}{\left( \frac{\lambda_{e,\text{max}}^{0.5}}{2} \right)}
\]

(8)

An efficient way of obtaining an estimate of the maximum eigenvalues of an element is to employ the Rayleigh quotient.
where $\phi_{iI}$ is the assumed highest mode eigenvector for the element.

Because the penalty element is modelling the traction between two contacting surfaces, it does not have any mass associated with itself. Thus, the unconstrained element eigenvalues for a zero mass element are infinity and the critical time step for a mesh with contact elements cannot be obtained from equation (7). However, the assembled problem does have finite eigenvalues and can be used to obtain the critical time step.

NODE-BASED STABILITY CRITERIA

At this point, the objective is to obtain an estimate for the critical time step based upon the assembled lumped nodal mass components $m_{iIjJ}$ and the nodal stiffness components $k_{iIjJ}$. To accomplish this, Gerschgorin's theorem is used to bound the maximum eigenvalue of the discretized contact problem.

Gerschgorin's circle theorem (e.g. Noble [6]) states that each eigenvalue $\lambda_{iI}$ of the $n \times n$ matrix $A_{iIjJ}$ satisfies

$$|A_{iIiI} - \lambda_{iI}| \leq \sum_{j \neq i} |A_{iIjJ}|$$

So the maximum eigenvalue is given by

$$\lambda_{n,\text{max}} \leq \max_{n}(\lambda_{iI})$$

(11)

A comparison between equations (6) and (11) shows that the former is based upon unassembled element information and the latter is based upon assembled nodal information.

By applying Gerschgorin's theorem, the time step $\Delta t_{iI}$ for each node belonging to a contact element can be expressed as

$$\Delta t_{iI} \leq \frac{2 (m_{iIiI})^{0.5}}{[K_{iIiI} + \sum_{j \neq i, j \neq I} |K_{iIjJ}|]^{0.5}}$$

(12)
The maximum time step for the nodes, $n_c$, involved in initial impact is determined from

$$\Delta t_{iI,\text{max}} = \max_{n_c} (\Delta t_{iI})$$

(13)

The critical time step for the problem, which consists of both impacting and non-impacting elements, is determined from

$$\Delta t_{\text{crit}} = \min (\Delta t_{e,\text{max}}, \Delta t_{iI,\text{max}})$$

(14)

A significant point to note is that the row sum in equation (12) only involves the elements that are in contact at the beginning of the impact simulation. Therefore, only the stiffness for these elements is calculated. Furthermore, since only the value of the row sum is needed, they are accumulated in a column matrix, which is referred to as a partially assembled stiffness matrix. The storage requirement for this matrix is significantly less than that of a completely assembled matrix.

### NUMERICAL EXAMPLES

A problem is presented to illustrate the use of the combined elemental and nodal stability criteria in application to impact.

The example is the longitudinal impact of elastic bars (Figure 1a). Dimensionless quantities were used. Both bars were assumed to have a length of 10.0 and a cross sectional area of 1.0. The density and stiffness of the bars were varied as explained below. Figure 1b shows the finite element mesh, which consists of 20 axial elements per bar with a contact element between them. The element stiffness of the left bar, $k_{LB}$, right bar, $k_{RB}$, and contact element, $k_{CE}$, were varied as shown in Table 1. The total mass of each element in the left bar, $m_{LB}$, and right bar, $m_{RB}$, were also varied. The values of the eigenvalue for each bar element were computed by Rayleigh's quotient, and the maximum eigenvalue, $\lambda_{e,\text{max}}$, is tabulated in Table 1. Next, a Rayleigh quotient for the contact element, $\lambda_{ce,\text{RQ}}$, was computed using the contact stiffness and the assembled nodal mass for each node at the ends of the contact element. Note, the mass contributions come only from the bar elements in contact; the contact element does not contribute to the mass. A Gerschgorin bound, $\lambda_{n,\text{max}}^G$, was the computed only for the contact element. Finally, the exact maximum eigenvalue, $\lambda_{E,\text{max}}$, of the assembled problem, equation (5), was obtained with the basic Jacobi solution method. For Case 1, in which the contact element stiffness is the same as the bar element stiffness, the Rayleigh quotient for the
contact element underestimates the exact value by 39 percent and thus would lead to an unstable time step; however the Gerschgorin bond overestimates by 21 percent and thus leads to a stable time step estimate. For Case 2, the contact stiffness was assumed to be ten times the bar element stiffness. Here it is seen that the Rayleigh quotient for the contact element only underestimates the exact value by 5 percent, and the Gerschgorin bound gives an overestimation of 5 percent. For the cases considered, the Gerschgorin bound always leads to a stable time step estimation, and the Rayleigh quotient always leads to an unstable time step prediction.

The impact of the two bars was simulated with the NEPTUNE code. The Young's modulus for both bars were taken to be 100, the density was 0.01. The left bar was assumed to have an initial velocity of 0.1 and the right bar was assumed to be at rest. The displacement, velocity, and force signatures during the impact event are shown in Figures 2 to 4.

CONCLUSIONS

An approach has been developed to determine the critical time step for problems involving impact using explicit time-marching integrators. The approach utilizes unassembled element information to obtain critical time steps estimates for elements outside the contact region and assembled nodal information to obtain critical time steps for nodes in the contact region. The approach is independent of element type.

ACKNOWLEDGEMENT

This work was performed under the auspices of the U. S. Department of Energy, Office of Technology Support Programs, under contract W-31-109-Eng-38.

REFERENCES


<table>
<thead>
<tr>
<th>Case</th>
<th>( RQ_{\text{e, max}} )</th>
<th>( RQ_{CE} )</th>
<th>( G_{n, \text{ max}} )</th>
<th>( \lambda_{\text{max}} )</th>
</tr>
</thead>
<tbody>
<tr>
<td>1. ( k_{LB} = 1, k_{RB} = 1 ) ( k_{CE} = 1 ) ( m_{LB} = 1, m_{RB} = 1 )</td>
<td>4.0</td>
<td>4.0</td>
<td>8.0</td>
<td>6.6</td>
</tr>
<tr>
<td>2. ( k_{LB} = 1, k_{RB} = 1 ) ( k_{CE} = 10 ) ( m_{LB} = 1, m_{RB} = 1 )</td>
<td>4.0</td>
<td>40.0</td>
<td>44.0</td>
<td>42.6</td>
</tr>
<tr>
<td>3. ( k_{LB} = 1, k_{RB} = 1 ) ( k_{CE} = 1 ) ( m_{LB} = 1, m_{RB} = 10 )</td>
<td>4.0</td>
<td>0.7</td>
<td>8.0</td>
<td>5.3</td>
</tr>
<tr>
<td>4. ( k_{LB} = 1, k_{RB} = 1 ) ( k_{CE} = 10 ) ( m_{LB} = 1, m_{RB} = 10 )</td>
<td>4.0</td>
<td>3.6</td>
<td>44.0</td>
<td>34.4</td>
</tr>
</tbody>
</table>
Figure 1. (a) Two impacting bars. (b) Finite element model of impacting bars.

Figure 2. Displacement signature of contact interface.
Figure 3. Velocity signature of contact interface.

Figure 4. History of contact interface force.