Comparison of implicit and explicit procedures

Abaqus/Standard is more efficient for solving smooth nonlinear problems; on the other hand, Abaqus/Explicit is the clear choice for a wave propagation analysis. There are, however, certain static or quasi-static problems that can be simulated well with either program. Typically, these are problems that usually would be solved with Abaqus/Standard but may have difficulty converging because of contact or material complexities, resulting in a large number of iterations. Such analyses are expensive in Abaqus/Standard because each iteration requires a large set of linear equations to be solved.

Comparison of implicit and explicit procedures

Whereas Abaqus/Standard must iterate to determine the solution to a nonlinear problem, Abaqus/Explicit determines the solution without iterating by explicitly advancing the kinematic state from the previous increment. Even though a given analysis may require a large number of time increments using the explicit method, the analysis can be more efficient in Abaqus/Explicit if the same analysis in Abaqus/Standard requires many iterations.

Another advantage of Abaqus/Explicit is that it requires much less disk space and memory than Abaqus/Standard for the same simulation. For problems in which the computational cost of the two programs may be comparable, the substantial disk space and memory savings of Abaqus/Explicit make it attractive.

Metal forming processes are generally considered quasi-static in nature. Inertial effects are assumed to be negligible.

To model such processes, two basic solution procedures are used: *implicit* and *explicit* methods.

- The implicit procedure is available in ABAQUS/Standard.
- The explicit procedure is available in ABAQUS/Explicit.

In both ABAQUS/Standard and ABAQUS/Explicit the nonlinear response is obtained *incrementally*.

Equilibrium

The discretized equilibrium equations for the finite element model are

$$P^N - I^N = M^{NM} \ddot{u}^M,$$

where

 P^N is the external force vector,

 I^N is the internal force vector (created by stresses in the elements), and $M^{NM}\ddot{u}^M$ is the force vector due to material inertia.

This equilibrium statement is quite general.

The "internal forces" are given by

$$I^N = \int_V \beta^N : \sigma \, dV \,,$$

CONTRACTOR AND A STATE OF A DESCRIPTION OF A

where

V is the current volume of the model,

 $\sigma(x)$ is the stress at a point currently located at position x, and $\beta^{N}(x)$ is the strain rate-displacement rate transformation defined from the interpolation assumption in the element,

$$\dot{\varepsilon} = \beta^N \dot{u}^N.$$

Note: throughout these notes, repeated superscripts imply summation over the degrees of freedom of the model.

Dynamic equilibrium implies that the d'Alembert forces,

 $M^{NM}\ddot{u}^{M},$

where M^{NM} is the mass matrix and \ddot{u}^{N} is the acceleration vector, are significant.

Static equilibrium means that the d'Alembert forces vary slowly, or are constant, with time (such as gravity or centrifugal loading). This implies $M^{NM}\ddot{u}^M \approx 0$.

Implicit Solution of Static Equilibrium

The implicit procedure is effective when the analysis can be performed in relatively few time (load) increments.

ABAQUS/Standard uses Newton's method to solve for static equilibrium. We assume we have an estimate of the solution at iteration (i), $u_{(i)}^N$, so that a Taylor series expansion about this estimate is

$$P^{N} - I^{N} + \left(\frac{\partial P^{N}}{\partial u^{M}} - \frac{\partial I^{N}}{\partial u^{M}}\right)c^{M} + \dots = 0.$$

We write this as

$$\boldsymbol{K}^{NM}\boldsymbol{c}^{M} = \boldsymbol{P}^{N} - \boldsymbol{I}^{N},$$

where

$$K^{NM} = \frac{\partial I^{N}}{\partial u^{M}} - \frac{\partial P^{N}}{\partial u^{M}}$$

is the system's tangent stiffness, or Jacobian, matrix, and c^N is the correction to the solution at degree of freedom N.

The incremental displacements (Δu) are updated by

$$\Delta u_{(i+1)}^{N} = \Delta u_{(i)}^{N} + c_{(i)}^{N}.$$

Iterations are repeated in each increment until convergence is achieved, which means that:

- Contact conditions are satisfied at every node (where applicable).
- Force equilibrium is attained at every node.
- Moment equilibrium is attained at every node (where applicable).
- Displacement corrections are small compared to incremental displacements.

ABAQUS/Standard uses automatic incrementation by default. The rate of convergence is monitored to determine appropriate time (load) increments.

- The increment size is increased if few iterations are required.
- The increment size is decreased if convergence is slow.
- If convergence cannot be obtained, the increment size is decreased and further attempts are made.

Explicit Solution of Dynamic Equilibrium

The explicit procedure performs the analysis using a large number of inexpensive, small, time (load) increments.

ABAQUS/Explicit solves for a state of dynamic equilibrium at the start of the current time increment (*t*):

$$M^{NM} \ddot{u}^N \big|_t = \left. (P^N - I^N) \right|_t.$$

In the explicit procedure, a diagonal mass matrix is used for efficiency. Thus, the nodal accelerations can be obtained easily:

$$\ddot{u}^{N}|_{t} = [M^{NM}]^{-1} \cdot (P^{M} - I^{M})|_{t}$$

The central difference integration rule is used to update the velocities and displacements:

$$\begin{split} \dot{u}^{N}\Big|_{t+\frac{\Delta t}{2}} &= \left. \dot{u}^{N} \right|_{t-\frac{\Delta t}{2}} + \left(\frac{\Delta t \Big|_{t+\Delta t} + \Delta t \Big|_{t}}{2} \right) \ddot{u}^{N} \Big|_{t} \,, \\ u^{N}\Big|_{t+\Delta t} &= \left. u^{N} \Big|_{t} + \Delta t \Big|_{t+\Delta t} \dot{u}^{N} \Big|_{t+\frac{\Delta t}{2}} \,. \end{split}$$

No iterations are required in the equation solver to update the accelerations, velocities, and displacements.

The central difference integration rule is only conditionally stable; that is the solution becomes unstable and diverges rapidly if the time increment is too big.

An estimate of the stable time increment is given by

$$\Delta t = \min\left(\frac{L^e}{c_d}\right),$$

where L^{e} is a characteristic element length and c_{d} is the dilatational wave speed of the material. For a linear elastic material,

$$c_d = \sqrt{\frac{\lambda + 2\mu}{\rho}},$$

where λ and μ are Lame's constants and ρ is the material density.

If we consider a finite element mesh of a single material (e.g., sheet metal blank), the stable time increment will be dictated by the smallest element in the mesh.

The explicit procedure is well-suited for high-speed (wave propagation) applications.

- The stable time increment is generally small for realistic meshes modeling real engineering materials.
- The solution of individual time increments is inexpensive.
- Fine resolution solutions of high-speed events can be obtained at relatively low cost.
- For steel, c = 5000 m/sec, a typical element size might be ~10 mm, so $\Delta t = 2 \times 10^{-6}$ sec. Thus, an event lasting 0.5 sec requires 250000 time increments.

Comparison of Implicit and Explicit Procedures for Metal Forming

Because we consider metal forming processes to be quasi-static, it would appear that the implicit static procedure is an appropriate choice for modeling purposes. However, especially for three-dimensional problems, the explicit procedure is generally more efficient computationally.

Model Size

- The cost of an explicit *increment* is approximately proportional to the model size.
- As model size increases, solution cost for the implicit procedure increases more steeply than for the explicit procedure.

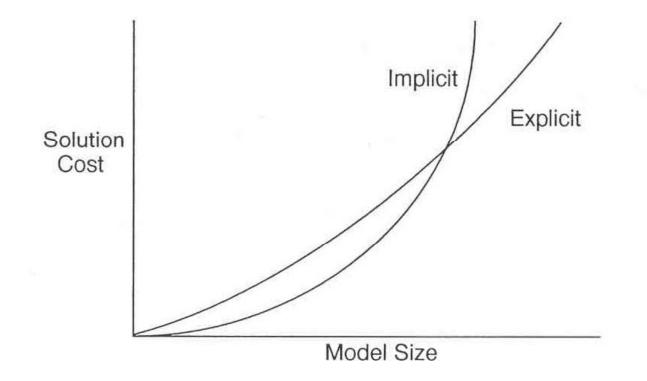


Figure 2–1. Comparison of Implicit and Explicit Solution Costs

Contact

Sheet metal forming processes are dominated by contact.

- The implicit procedure in ABAQUS/Standard uses constraints to enforce contact conditions.
 - The solver must iterate to satisfy all contact conditions.
 - Cutback of the time increment will occur if contact conditions cannot be satisfied in a reasonable number of iterations.
 - Three-dimensional models have a large number of possible contact points, and changing contact conditions can result in extremely small time increments or convergence difficulties.

- The explicit procedure in ABAQUS/Explicit offers two methods to enforce contact conditions: a kinematic predictor/corrector algorithm and a penalty method.
 - No iterations are required to enforce contact conditions.
 - The time increment size is independent of the number of contact points and the complexity of the contact conditions for the kinematic method.
 - The penalty method can affect the size of the stable time increment.

Local Instabilities (Wrinkling)

The nature of drawing processes induces compressive stresses, which can lead to wrinkling.

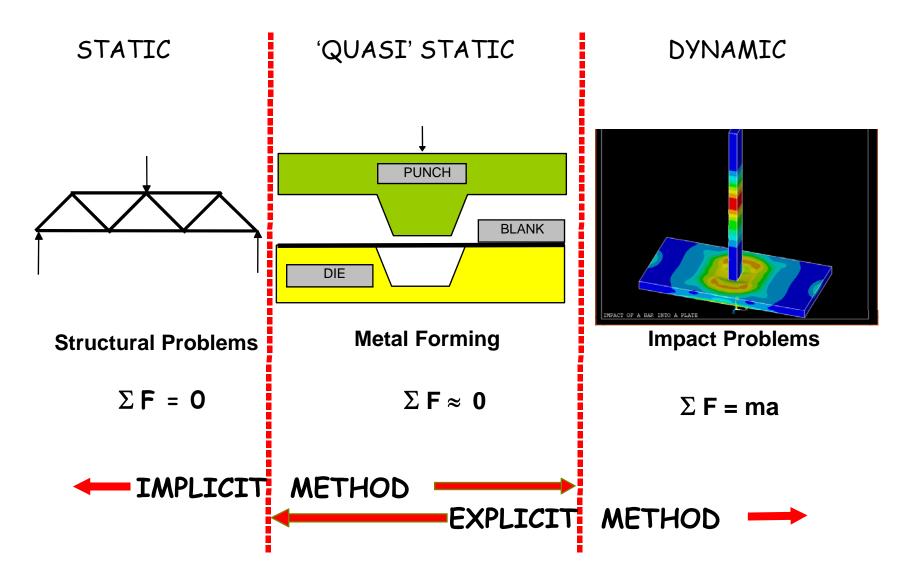
- Wrinkling can cause severe convergence difficulties in the implicit procedure.
 - The implicit procedure works best for smooth nonlinear response.
 - Wrinkling represents a sudden buckling discontinuity in the nonlinear response.
 - Substantial cutback of the time increment size may be required.
 - Convergence may not be possible in some cases.

- · Wrinkling causes no difficulties in the explicit procedure.
 - Accelerations, velocities, and displacements are obtained without regard to the severity of the discontinuity.
 - Inertia prevents unchecked growth of the instability.

The explicit method also has limitations.

- Generally, only first-order, reduced-integration elements can be used. This is a severe limitation if continuum elements must be used to represent bending.
- Since the method solves problems by propagating disturbances as waves (which travel from elements to their nearest neighbors in each time increment), it can be extremely slow for very simple stages, such as springback calculations in sheet forming.

The emergence of high-speed vector hardware, along with highly vectorized finite element software, has made explicit methods much more attractive.



Implicit Time Integration:

•Inertia effects ([C] and [M]) are typically not included

•Average acceleration - displacements evaluated at time t+Dt:

$$\left\{ u_{t+\Delta t} \right\} = \left[K \right]^{-1} \left\{ F_{t+\Delta t}^{a} \right\}$$

Linear Problems:

>Unconditionally stable when [K] is linear

≻Large time steps can be taken

Nonlinear problems:

Solution obtained using a series of linear approximations (Newton-Raphson)

>Requires inversion of nonlinear stiffness matrix [K]

- >Small iterative time steps are required to achieve convergence
- >Convergence is not guaranteed for highly nonlinear problems

Comparison of Implicit and Explicit Methods <u>Explicit Time Integration</u>:

Central difference method used - accelerations evaluated at time t:

Where $\{F_t^{ext}\}$ is the applied external and body force vector,

 $\{F_t^{int}\}$ is the internal force vector which is given by:

$$\{\mathbf{a}_{t}\} = [\mathbf{M}]^{-1} \left([\mathbf{F}_{t}^{\text{ext}}] - [\mathbf{F}_{t}^{\text{int}}] \right)$$
$$F^{\text{int}} = \Sigma \left(\int_{\Omega} B^{T} \boldsymbol{\sigma}_{n} d\Omega + F^{hg} \right) + F^{\text{contact}}$$

- F^{hg} is the hourglass resistance force (see ELEMENTS Chapter) and F^{cont} is the contact force.
- The velocities and displacements are then evaluated:

$$\{v_{t+\Delta t/2}\} = \{v_{t-\Delta t/2}\} + \{a_t\}\Delta t_t$$
$$\{u_{t+\Delta t}\} = \{u_t\} + \{v_{t+\Delta t/2}\}\Delta t_{t+\Delta t/2}$$

where $\Delta t_{t+\Delta t/2} = .5(\Delta t_{t} + \Delta t_{t+\Delta t})$ and $\Delta t_{t-\Delta t/2} = .5(\Delta t_{t} - \Delta t_{t+\Delta t})$

Comparison of Implicit and Explicit Methods <u>Explicit Time Integration</u>:

The geometry is updated by adding the displacement increments to the initial geometry $\{x_o\}$:

$$\{x_{t+\Delta t}\} = \{x_o\} + \{u_{t+\Delta t}\}$$

- Nonlinear problems:
- > Lumped mass matrix required for simple inversion
- Equations become uncoupled and can be solved for directly (explicitly)
- No inversion of stiffness matrix is required. All nonlinearities (including contact) are included in the internal force vector.
- > Major computational expense is in calculating the internal forces.
- > No convergence checks are needed
- Very small time steps are required to maintain stability limit

Stability Limit

Implicit Time Integration:

For linear problems, the time step can be arbitrarily large (always stable)

For nonlinear problems, time step size may become small due to convergence difficulties

Explicit Time Integration:

Only stable if time step size is smaller than critical time step size

$$\Delta t \le \Delta t^{crit} = \frac{2}{\omega_{\max}}$$

Where w_{max} = largest natural circular frequency Due to this very small time step size, explicit is useful only for very short transients

Critical Time Step Size

Critical time step size of a rod

- Natural frequency:

$$\omega_{\max} = 2\frac{c}{l}$$
 with $c = \sqrt{\frac{E}{\rho}}$ (wave propagation velocity)

Critical time step:

$$\Delta t = \frac{l}{c}$$

- Courant-Friedrichs-Levy-criterion
- Δt is the time needed of the wave to propagate through the rod of length /

Note: The critical time step size for explicit time integration depends on element length and material properties (sonic speed).

ABAQUS/EXPLICIT Time Step Size

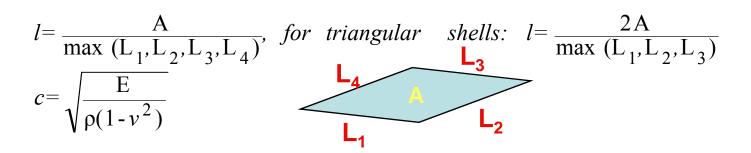
ABAQUS/EXPLICIT checks all elements when calculating the required time step.

The characteristic length $\underline{/}$ and the wave propagation velocity \underline{c} are dependent on element type:

Beam elements:

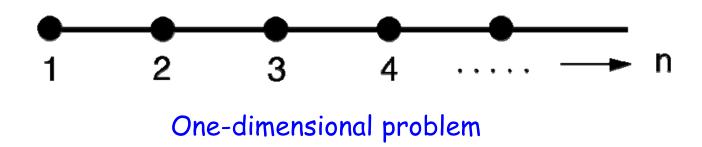
l = length of the element $c = \sqrt{\frac{E}{0}}$

Shell elements:



ABAQUS/EXPLICIT Time Step Size

- The concept of a stable time increment is explained easily by considering a one-dimensional problem.



- The stable time increment is the minimum time that a dilatational wave takes to move across any element in the model.
- A dilatational wave consists of volume expansion and contraction.

ABAQUS/EXPLICIT Time Step Size

- Thus, the stable time increment can be expressed as

$$\Delta t = \frac{l}{c}$$

- Decreasing L and/or increasing c will reduce the size of the stable time increment.

- Decreasing element dimensions reduces L.
- Increasing material stiffness increases c.
- Decreasing material compressibility increases c.
- Decreasing material density increases c.
- ABAQUS/Explicit monitors the finite element model throughout the analysis to determine a stable time increment.

- **Implicit**: Solve for $t + \Delta t$ using state at t and $t + \Delta t$
 - don't know state at $t+\Delta t$ yet
 - Newton-Raphson method used typically ABAQUS/Standard, ANSYS, MARC,...
 - take initial guess and **iterate** to convergence
 - end up solving "linear-like" equation for each iteration: $\mathbf{K}\mathbf{u} = \mathbf{F}$
 - very accurate
 - can use relatively large time steps

Implicit Time Integration (used by ANSYS) -

- •Finite Element method used
- Average acceleration calculated
- Displacements evaluated
- •Always stable but small time steps needed to capture transient response
- •Non-linear materials can be used to solve static problems
- •Can solve non-linear (transient) problems...
- •...but only for linear material properties
- Best for static or 'quasi' static problems

- **Explicit**: Solve for $t+\Delta t$ using state at t
 - know state at t so can calculate \mathbf{K}_t
 - solve directly for incremental displacements:

 $\mathbf{K}_t \Delta \mathbf{u} = \Delta \mathbf{F}$

- no iteration
- no convergence check
- usually used in purpose written codes
- method is very robust
- must use very small time steps $(x10 \rightarrow x1000)$

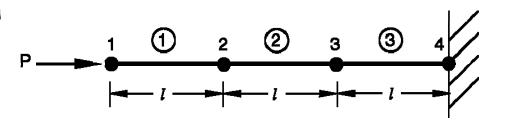
Explicit Time Integration (used by LS Dyna)

- Central Difference method used
- Accelerations (and stresses) evaluated
- Accelerations -> velocities -> displacements
- Small time steps required to maintain stability
- •Can solve non-linear problems for non-linear materials
- Best for dynamic problems

Overview of the Explicit Dynamics Procedure

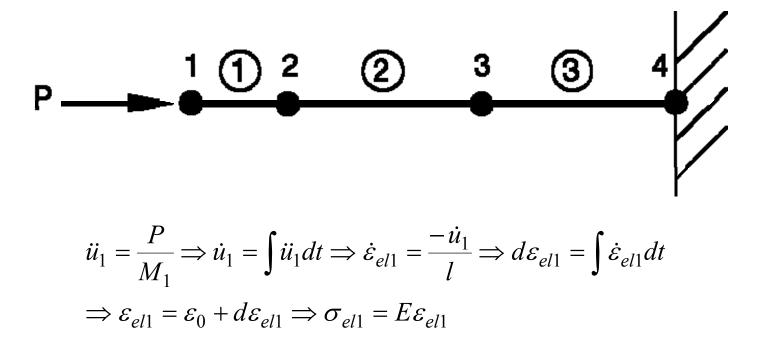
Stress wave propagation

- This stress wave propagation example illustrates how the explicit dynamics solution procedure works without iterating or solving sets of linear equations.
- We consider the propagation of a stress wave along a rod modeled with three elements. We study the state of the rod as we increment through time.
 - Mass is lumped at the nodes.



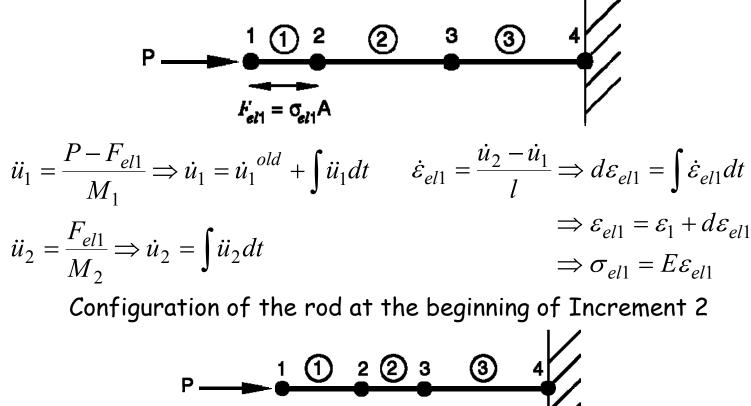
Initial configuration of a rod with a concentrated load, P, at the free end

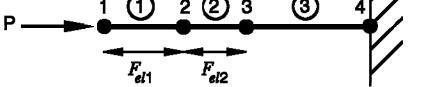
Overview of the Explicit Dynamics Procedure



Configuration at the end of Increment 1

Overview of the Explicit Dynamics Procedure





Configuration of the rod at the beginning of Increment 3

Explicit Dynamics method

$$M \cdot \ddot{U} + K \cdot U = F$$

 $\dot{U}(t_{n+(1/2)}) = \frac{U(t_{n+1}) - U(t_n)}{\Delta t},$
 $\ddot{U}(t_n) = \frac{\dot{U}(t_{n+(1/2)}) - \dot{U}(t_{n-(1/2)})}{\Delta t} = \frac{U(t_{n+1}) - 2U(t_n) + U(t_{n-1})}{(\Delta t)^2},$

Errors are of the order $\mathcal{O}((\Delta t)^2)$ for time steps $\Delta t \rightarrow 0$,

$$\begin{split} M \, \ddot{U}(t_n) &= F(t_n) - K \, U(t_n) \\ \dot{U}(t_{n+(1/2)}) &= \dot{U}(t_{n-(1/2)}) + \Delta t \, \ddot{U}(t_n) \qquad U(t_0) = U_0 \\ U(t_{n+1}) &= U(t_n) + \Delta t \, \dot{U}(t_{n+(1/2)}), \qquad \dot{U}(t_{0-(1/2)}) = \dot{U}_0 \end{split}$$

Implicit Dynamics method

$$\begin{split} M \ddot{U}(t_{n+1}) + (1+\alpha)K U(t_{n+1}) - \alpha K U(t_n) &= F(t_{n+1} + \alpha \Delta t) \quad (*) \\ U(t_{n+1}) &= U(t_n) + \Delta t \dot{U}(t_n) + \frac{1}{2} (\Delta t)^2 [(1-2\beta) \ddot{U}(t_n) + 2\beta \ddot{U}(t_{n+1})], \\ \dot{U}(t_{n+1}) &= \dot{U}(t_n) + \Delta t [(1-\gamma) \ddot{U}(t_n) + \gamma \ddot{U}(t_{n+1})], \end{split}$$

$$\alpha \in [-1/3, 0], \qquad \beta = (1-\alpha)^2/4, \qquad \gamma = (1-2\alpha)/2$$

 $U(t_0) = U_0$ $\dot{U}(t_0) = \dot{U}_0$ $\ddot{U}(t_0)$ from eq.*with $\alpha = 0, n = -1,$