Rock Mechanics Seminar Series 2010

Bonus 1: Principles of Numerical Solution

FEM mesh (2D for simplicity)

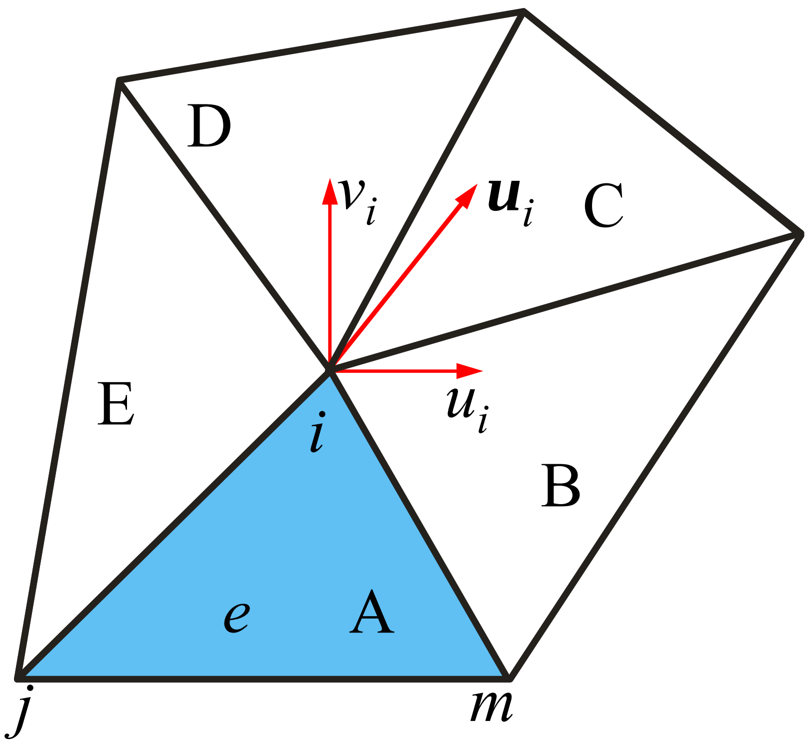


Figure shows part of our mesh, with some elements surrounding node i. (On figure elements A, B, C, D, E.) Think triangles, but ”any” shapes allowed. General element is called e, with nodes i, j, and m.

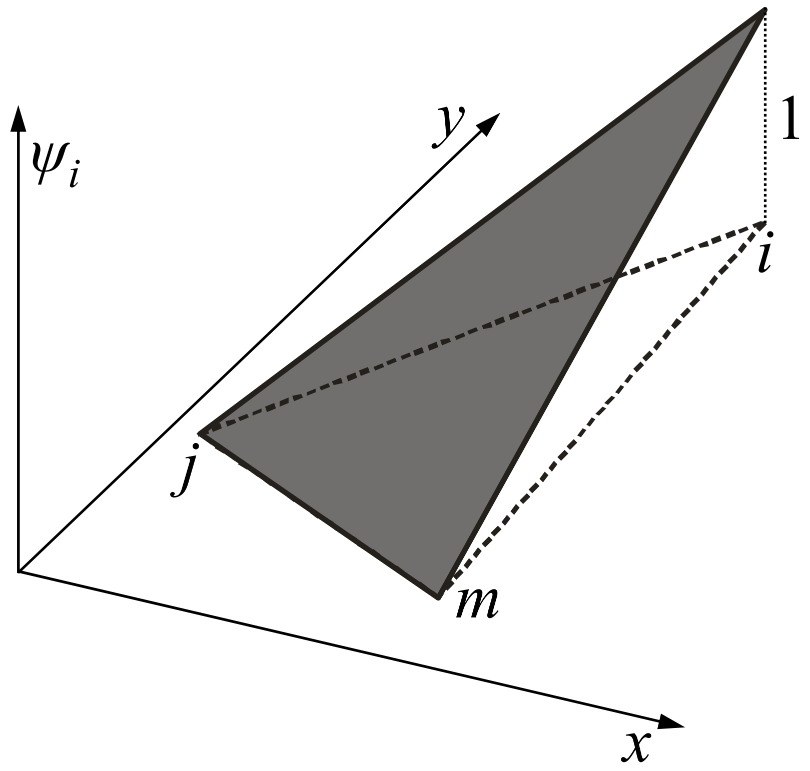
Displacement in node i: $\mathbf{u}_i = (u_i, v_i)^{\mathrm{T}}$

When some force is applied to e, a point $\boldsymbol{x}$ in e will have a displacement

$$\mathbf{u}(\boldsymbol{x}) = \{u(\boldsymbol{x}), v(\boldsymbol{x})\}^{\mathrm{T}}.$$

In our scheme we approximate **u** with: $\mathbf{u} \approx \hat{\mathbf{u}} = \sum_{k \in e} \psi_k \mathbf{u}_k^e = \mathbf{\Psi} \mathbf{u}^e$

Shape function



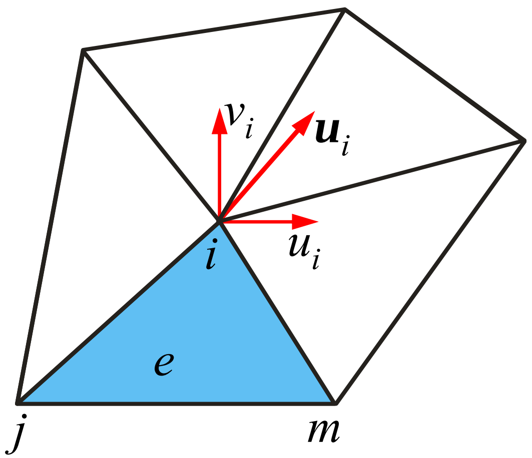
$\boldsymbol{\psi}$ is called the *shape function*.
Elements with a linear shape function are called linear elements, and ψ_i is typically as on the figure, i.e.
ψ_i is 1 in node *i*, and 0 in the other nodes.

$$\mathbf{u} \approx \hat{\mathbf{u}} = \sum_{k \in e} \psi_k \mathbf{u}_k^e = \boldsymbol{\Psi}\mathbf{u}^e$$

then means that $\hat{\mathbf{u}}$ is equal to $\mathbf{u}_k$ in node k ($k = i, j, m$),
and some interpolated value in the interior of the element.

$\hat{\mathbf{u}}$ is hence a linear approximation to $\mathbf{u}$ within the element.

Computing strains



We use the expression we derived earlier:

$$\boldsymbol{\varepsilon} = \begin{Bmatrix} \varepsilon_x \\ \varepsilon_y \\ \Gamma_{xy} \end{Bmatrix} = \begin{Bmatrix} \dfrac{\partial u}{\partial x} \\ \dfrac{\partial v}{\partial y} \\ \dfrac{\partial u}{\partial y} + \dfrac{\partial v}{\partial x} \end{Bmatrix} = \underbrace{\begin{bmatrix} \dfrac{\partial}{\partial x} & 0 \\ 0 & \dfrac{\partial}{\partial y} \\ \dfrac{\partial}{\partial y} & \dfrac{\partial}{\partial x} \end{bmatrix}}_{\mathbf{S}} \begin{bmatrix} u \\ v \end{bmatrix}$$

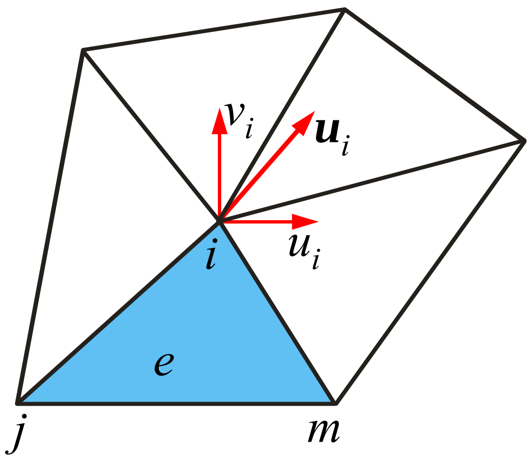
$$\Rightarrow \boldsymbol{\varepsilon} = \mathbf{S}\mathbf{u}$$

With $\mathbf{B} = \mathbf{S}\boldsymbol{\Psi}$ $\quad \boldsymbol{\varepsilon} \approx \hat{\boldsymbol{\varepsilon}} = \mathbf{B}\mathbf{u}$

is an expression for the nodal strains.

Note: I'm partially using the notation of Zienkiewicz, which may sometimes be conflicting with notation from prev. seminars. Sorry, but it wasn't easy to be consistent.

Stresses and nodal forces



For simplicity I omit initial / static stresses and assume isotropic material.
Then we expressed stress by strain as:

$$\boldsymbol{\sigma} = \mathbf{C}\boldsymbol{\varepsilon}$$

where

$$\mathbf{C} = \frac{E}{1-\nu^2}\begin{Bmatrix} 1 & \nu & 0 \\ \nu & 1 & 0 \\ 0 & 0 & \dfrac{1-\nu}{2} \end{Bmatrix}$$

A force $\mathbf{q}^e$ acts on e, nodal forces $\quad \mathbf{q}^e = \begin{Bmatrix} \mathbf{q}_i^e \\ \mathbf{q}_j^e \\ \mathbf{q}_m^e \end{Bmatrix}$

In addition body forces $\mathbf{b}$ act on the material (constant)

Virtual work

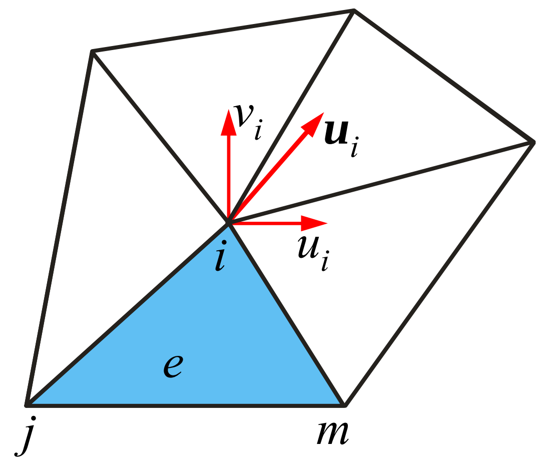
Impose virtual (infinitesimal) nodal displacements on element.
Equate external & internal work done by the applied forces & stresses during that displacement (std. FEM approach).

The nodal virtual displacement is $\delta\mathbf{u}^e$.
Within the element, $\delta\mathbf{u} = \mathbf{\Psi}\delta\mathbf{u}^e$, and $\delta\boldsymbol{\varepsilon} = \mathbf{B}\delta\mathbf{u}^e$.

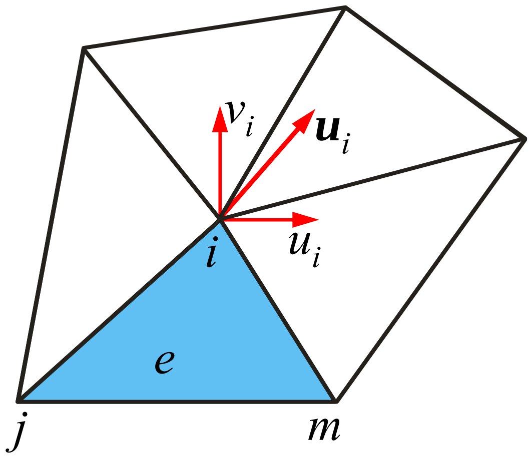
The work done by the nodal forces: $(\delta\mathbf{u}^e)^{\mathrm{T}}\mathbf{q}^e$ (distance x force)
Internal work per unit volume by stresses & body force:

$$\delta\boldsymbol{\varepsilon}^{\mathrm{T}}\boldsymbol{\sigma} - \delta\mathbf{u}^{\mathrm{T}}\mathbf{b}$$

$$\Leftrightarrow \delta\mathbf{u}^{\mathrm{T}}(\mathbf{B}^{\mathrm{T}}\boldsymbol{\sigma} - \mathbf{\Psi}^{\mathrm{T}}\mathbf{b})$$



Virtual work



$$\delta\boldsymbol{\varepsilon}^{\mathrm{T}}\boldsymbol{\sigma} - \delta\mathbf{u}^{\mathrm{T}}\mathbf{b}$$

$$\Leftrightarrow \delta\mathbf{u}^{\mathrm{T}}(\mathbf{B}^{\mathrm{T}}\boldsymbol{\sigma} - \boldsymbol{\Psi}^{\mathrm{T}}\mathbf{b})$$

Let ext. forces = int. forces over the elem. volume:

$$\delta(\mathbf{u}^e)^{\mathrm{T}}\mathbf{q}^e = \delta(\mathbf{u}^e)^{\mathrm{T}}\left[\int_{V^e}\mathbf{B}^{\mathrm{T}}\boldsymbol{\sigma}\,dV - \int_{V^e}\boldsymbol{\Psi}^{\mathrm{T}}\mathbf{b}\,dV\right]$$

As this is valid for all virtual displacements $\delta\mathbf{u}^e$:

$$\mathbf{q}^e = \int_{V^e}\mathbf{B}^{\mathrm{T}}\boldsymbol{\sigma}\,dV - \int_{V^e}\boldsymbol{\Psi}^{\mathrm{T}}\mathbf{b}\,dV$$

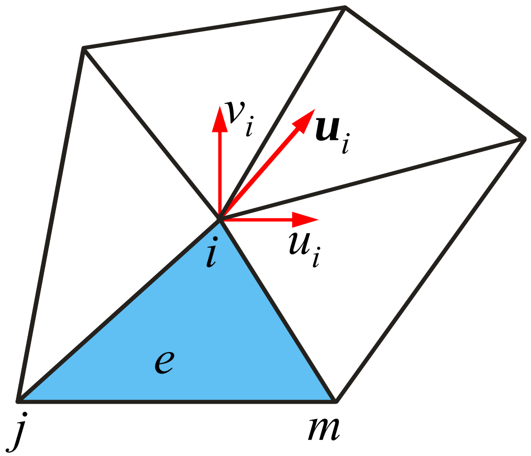
Recall $\boldsymbol{\sigma} = \mathbf{C}\boldsymbol{\varepsilon}$ and $\hat{\boldsymbol{\varepsilon}} = \mathbf{B}\mathbf{u}$

Hence $\mathbf{B}^{\mathrm{T}}\boldsymbol{\sigma} = \mathbf{B}^{\mathrm{T}}\mathbf{C}\boldsymbol{\varepsilon} = \mathbf{B}^{\mathrm{T}}\mathbf{C}\mathbf{B}\mathbf{u}$

$$\int_{V^e}\mathbf{B}^{\mathrm{T}}\boldsymbol{\sigma}\,dV = \int_{V^e}\mathbf{B}^{\mathrm{T}}\mathbf{C}\mathbf{B}\mathbf{u}^e\,dV \overset{\text{def}}{=} \mathbf{K}^e\mathbf{u}^e$$

$$-\int_{V^e}\boldsymbol{\Psi}^{\mathrm{T}}\mathbf{b}\,dV \overset{\text{def}}{=} \mathbf{f}^e$$

Virtual work



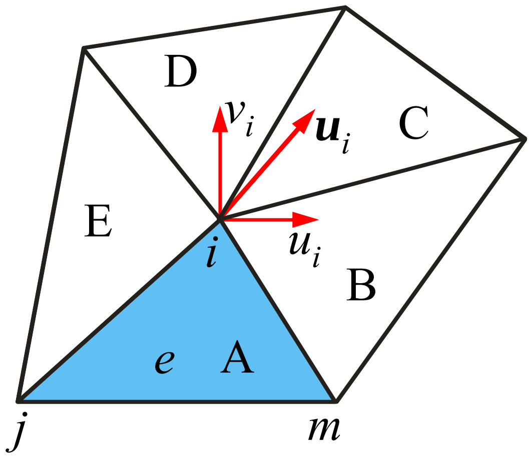
Summing up:

$$\mathbf{q}^e = \mathbf{K}^e\mathbf{u}^e + \mathbf{f}^e$$

$\mathbf{K}^e$ is the element stiffness matrix:

$$\mathbf{K}^e\mathbf{u}^e = \begin{Bmatrix} \mathbf{K}_{11} & \mathbf{K}_{12} & \cdots & \mathbf{K}_{1m} \\ \mathbf{K}_{21} & \mathbf{K}_{22} & \cdots & \mathbf{K}_{2m} \\ \vdots & & & \vdots \\ \mathbf{K}_{m1} & \mathbf{K}_{m2} & \cdots & \mathbf{K}_{mm} \end{Bmatrix} \begin{Bmatrix} \mathbf{u}_1 \\ \mathbf{u}_2 \\ \vdots \\ \mathbf{u}_m \end{Bmatrix}$$

Assembly



The structure we're studying (e.g. our reservoir) is generally also loaded by external forces **r**,
$\mathbf{r} = \{\mathbf{r}_1, \mathbf{r}_2, \ldots, \mathbf{r}_n\}^T$,
applied at the nodes in addition to the distributed loads applied to individual elements.

Equilibrium of node i requires that $\mathbf{r}_i$ is equal to the sum of forces contributed by the elements with node i as common point.

$$\mathbf{r}_i = \sum_{e=1}^{n_e} \mathbf{q}_i^e$$

(Obviously most of the terms vanish)

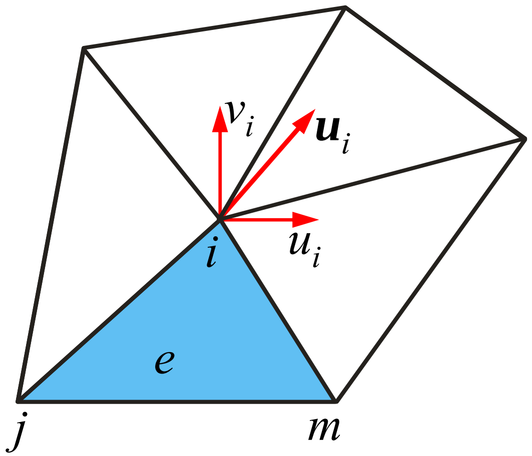
Example in figure: $\mathbf{r}_i = \mathbf{q}_i^A + \mathbf{q}_i^B + \mathbf{q}_i^C + \mathbf{q}_i^D + \mathbf{q}_i^E$

($\mathbf{q}_i^e$ is the force contributed to node i by element e)

Then:

$$\mathbf{r}_i = \left(\sum_{e=1}^{n_e} \mathbf{K}_{i1}^e\right)\mathbf{u}_1 + \left(\sum_{e=1}^{n_e} \mathbf{K}_{i2}^e\right)\mathbf{u}_2 + \ldots + \left(\sum_{e=1}^{n_e} \mathbf{K}_{im}^e\right)\mathbf{u}_{n_i} + \sum_{e=1}^{n_e} \mathbf{f}_i^e$$

Assembly



To get a global system to solve, we now assemble all the contributions from all the elements:

$$\mathbf{Ku} = \mathbf{r} - \mathbf{f}$$

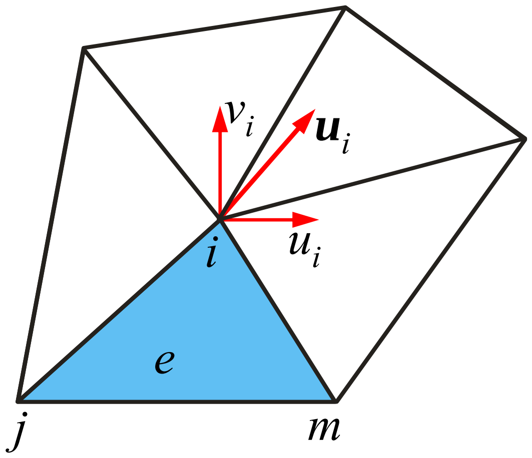
where: $$\mathbf{K}_{ij} = \sum_{e=1}^{n_e} \mathbf{K}_{ij}^{e}$$

$$\mathbf{f}_i = \sum_{e=1}^{n_e} \mathbf{f}_i^{e}$$

Tip:
To really understand the assembly procedure, try to program it!

This is called the stiffness formulation. The alternative, compliance formulation is also much used.
(System can be solved with error tolerance on stress, strain, or displacements.)

Short cuts



We have omitted (essential) parts of procedure

- Boundary conditions
  - Displacement BCs easily handled, other BCs require some computations
- Prior to element computations, every element is transformed to ”standard elements”
- We only considered linear elements with nodes on corners. In practice higher order elements with additional nodes (”integration points”) are often used.
- Triangular (or pyramids in 3D) are the simplest elements. ”Any” number of faces are permitted, typically chosen tailored to the problem.

Elasto-plastic solution proc. in Visage

After assembly of $\mathbf{K}$, solve for displacements $\mathbf{u}$ and $\Delta\mathbf{u}$.

i. Calculate strains: $\Delta\boldsymbol{\varepsilon} = \mathbf{B}\Delta\mathbf{u}$.

ii. Assume elasticity, calculate stress: $\Delta\boldsymbol{\sigma} = \mathbf{C}\Delta\boldsymbol{\varepsilon}$
Update $\boldsymbol{\sigma} = \boldsymbol{\sigma} + \Delta\boldsymbol{\sigma}$

iii. Compute stress invariants J_1, J_2, J_3.
(Typically p, q, and Lode angle (defined in a while))

iv. Use pre-selected failure criterion $F(J_1, J_2, J_3)$.
   a. If $F < 0$ at all integration points (FEM-nodes) the material behaves elastically.
   b. If $F \geq 0$ at any FEM-node, the yield criterion has been violated. Excess stresses exist, which must be redistributed into neighbouring FEM-nodes.
   (Expansion of yield surface?)

Elasto-plastic solution proc. in Visage

To reach a stationary equilibrium state, we solve for a time-dependent strain development when forces / stresses are applied in small increments with time increments Δt.
(Reminder: plastic strain may change with time even though the applied stress doesn't change.)

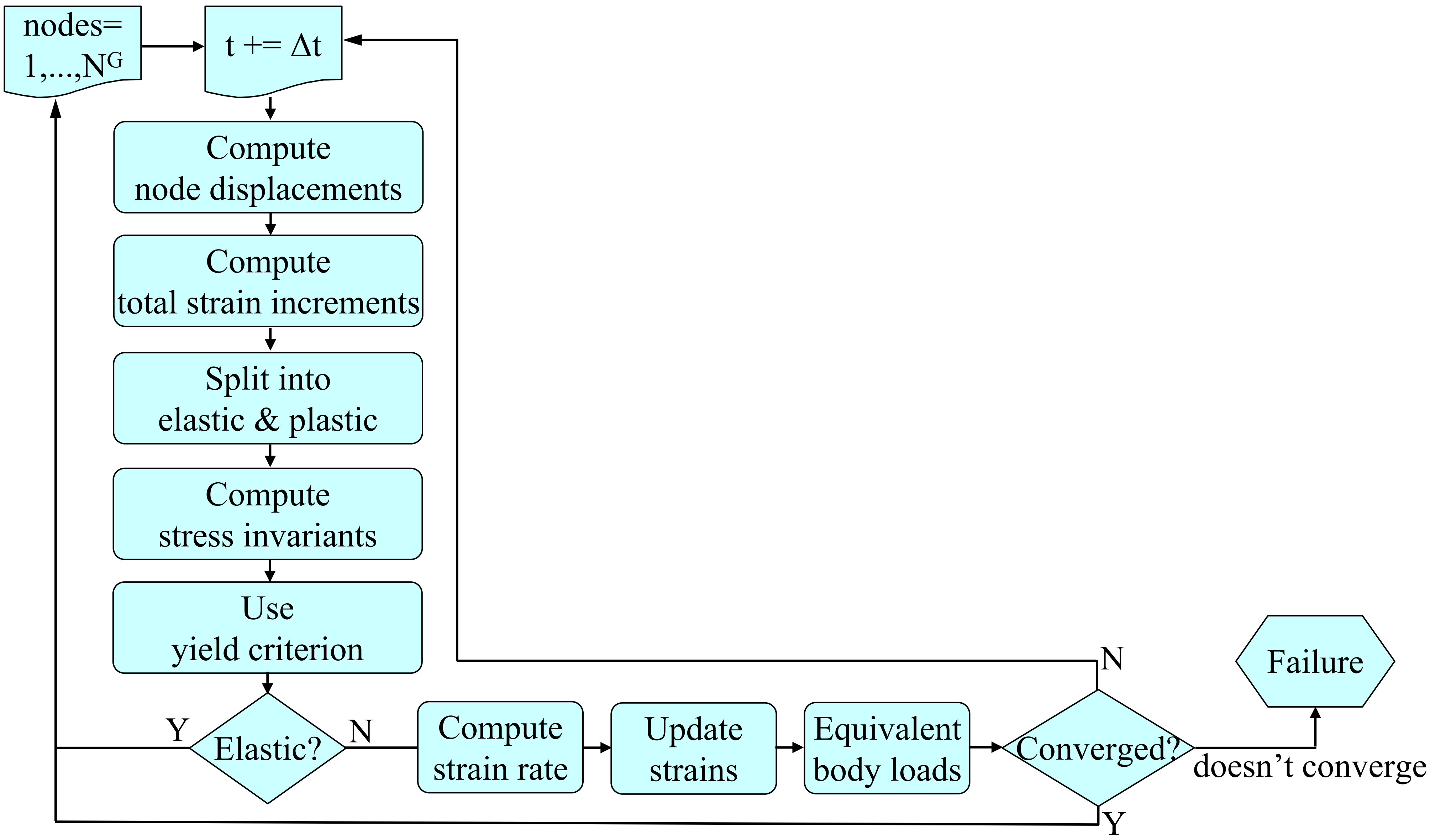
Total strain is divided into elastic and plastic: $\Delta\boldsymbol{\varepsilon} = \Delta\boldsymbol{\varepsilon}^e + \Delta\boldsymbol{\varepsilon}^p$
The plastic strain rate is:

$$\dot{\varepsilon}^p = f(F)\frac{\partial g}{\partial \sigma}$$ (a little simplified compared to actual expr.)

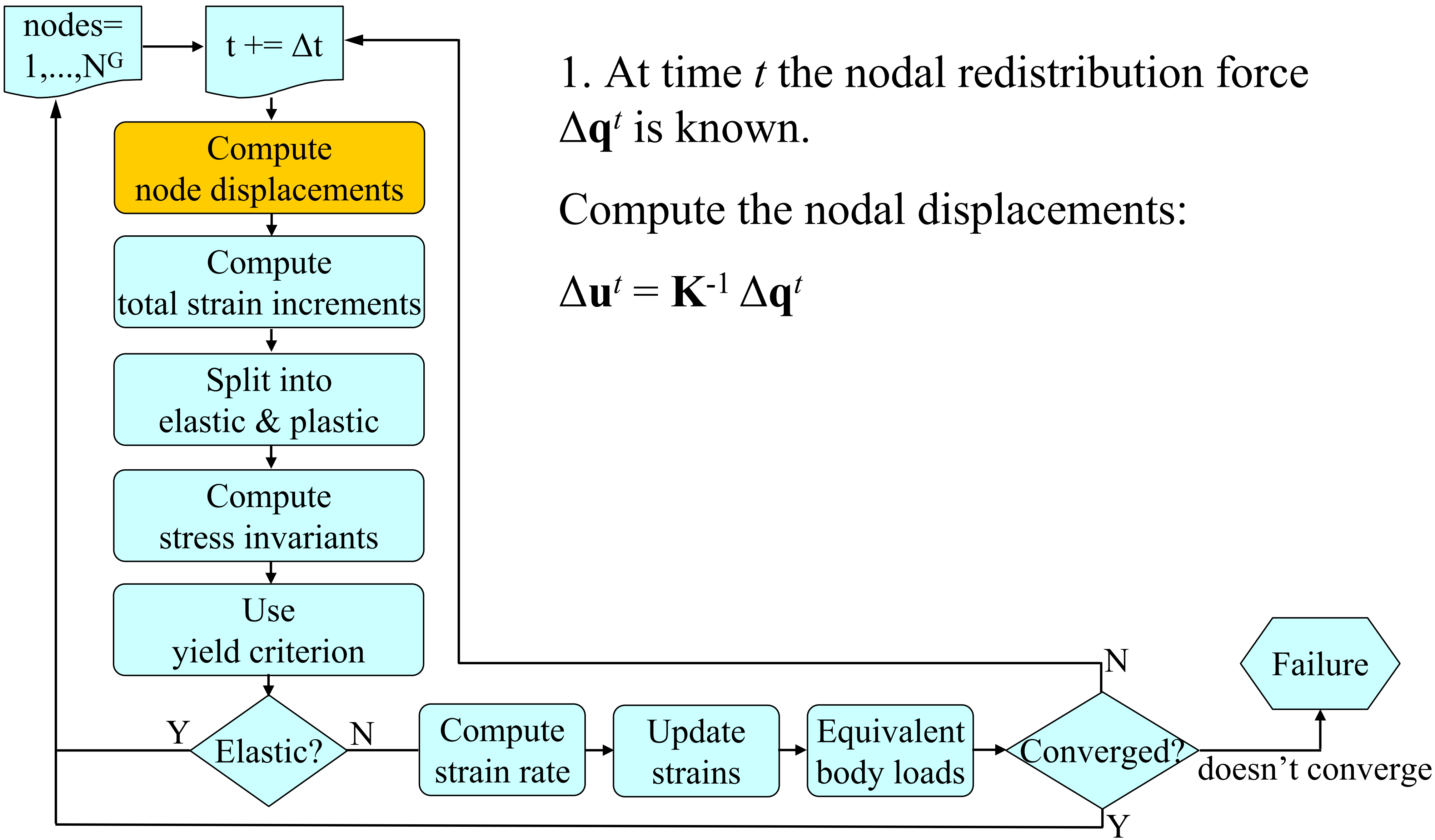
g is the visco-plastic potential

The complete flow chart is then (next slide):

Visco-plasticity flow chart



Visco-plasticity flow chart

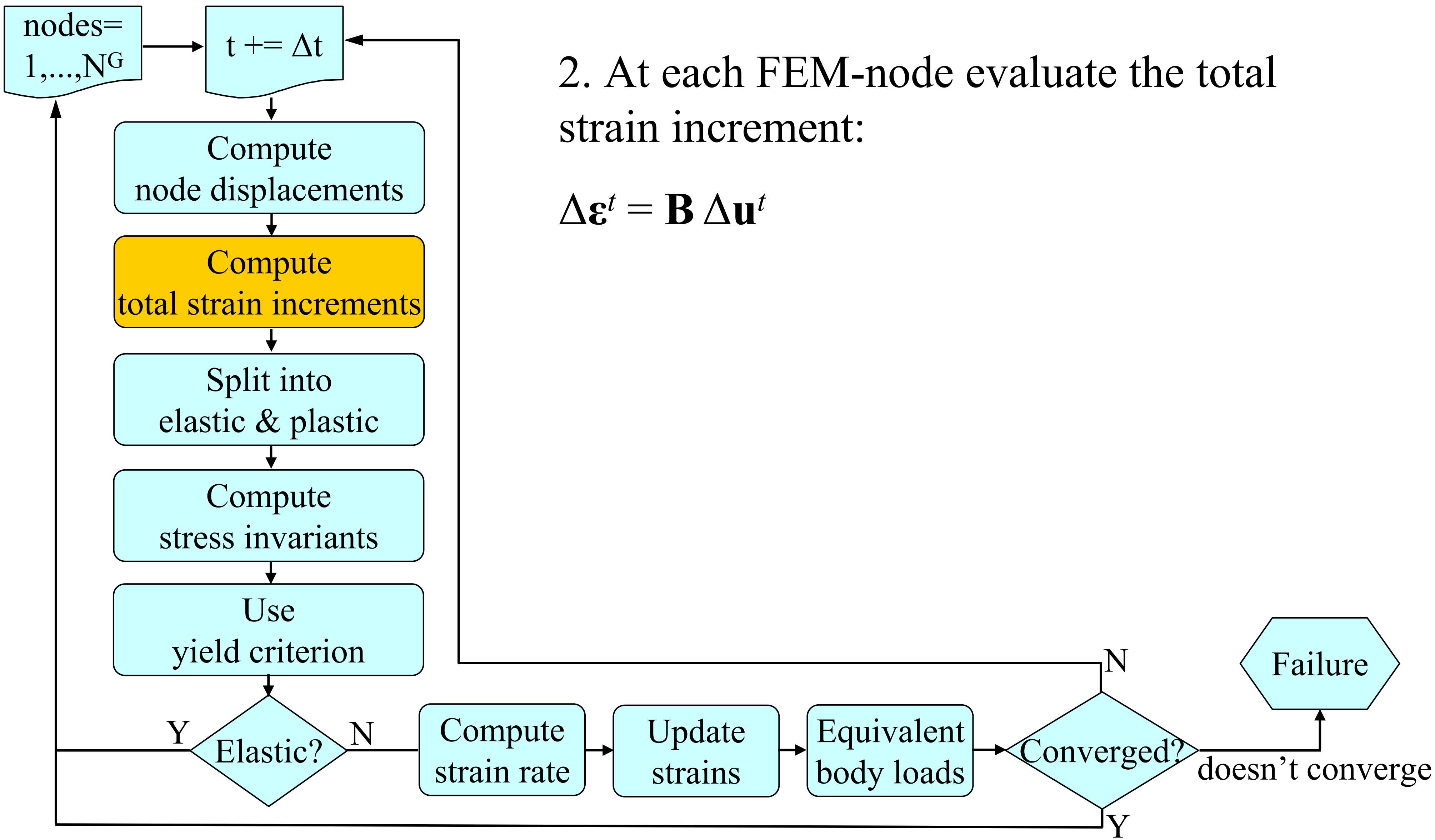


1. At time t the nodal redistribution force $\Delta \mathbf{q}^t$ is known.

Compute the nodal displacements:

$$\Delta \mathbf{u}^t = \mathbf{K}^{-1} \Delta \mathbf{q}^t$$

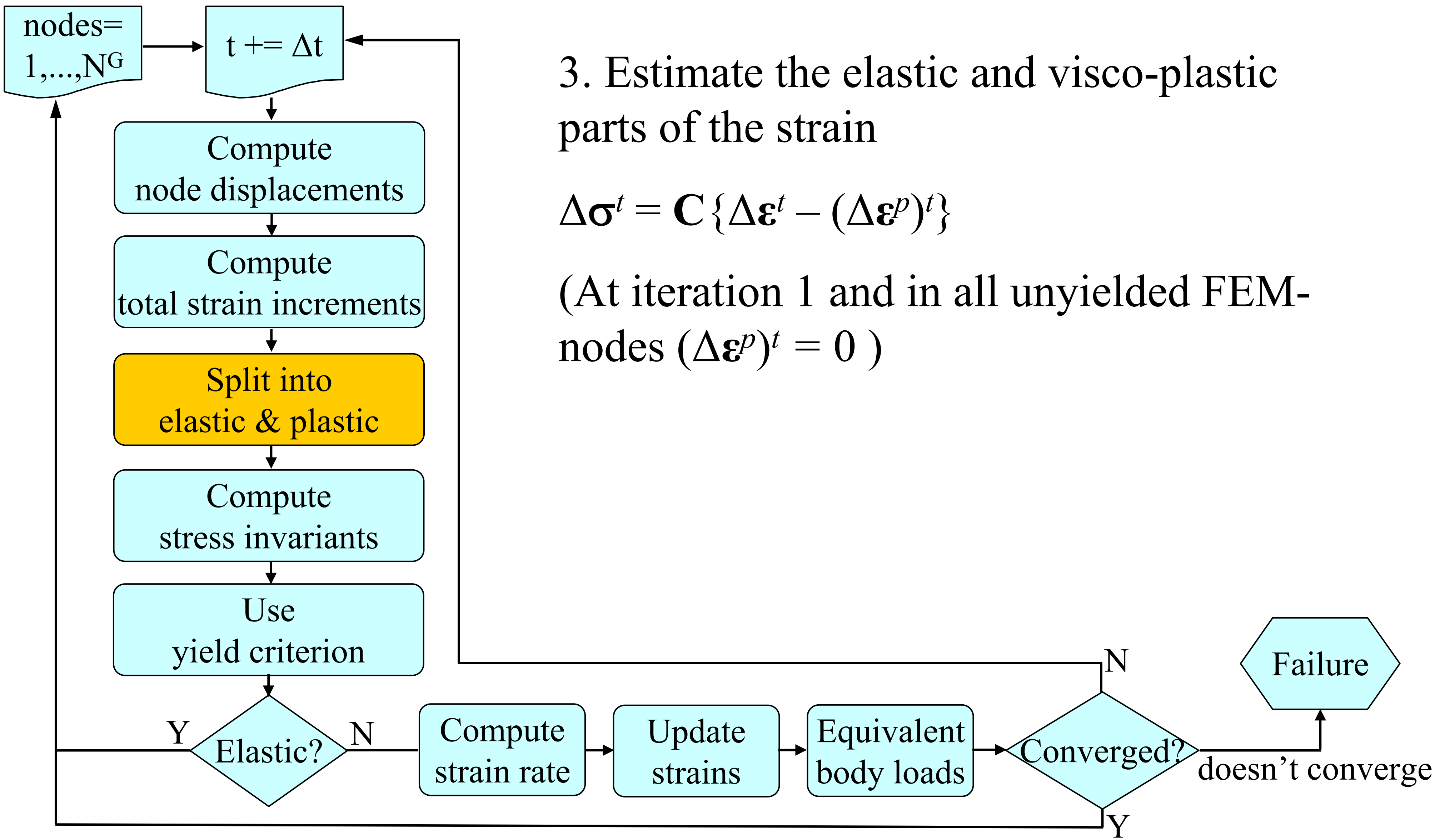
Visco-plasticity flow chart



2. At each FEM-node evaluate the total strain increment:

$$\Delta\boldsymbol{\varepsilon}^t = \mathbf{B}\,\Delta\mathbf{u}^t$$

Visco-plasticity flow chart

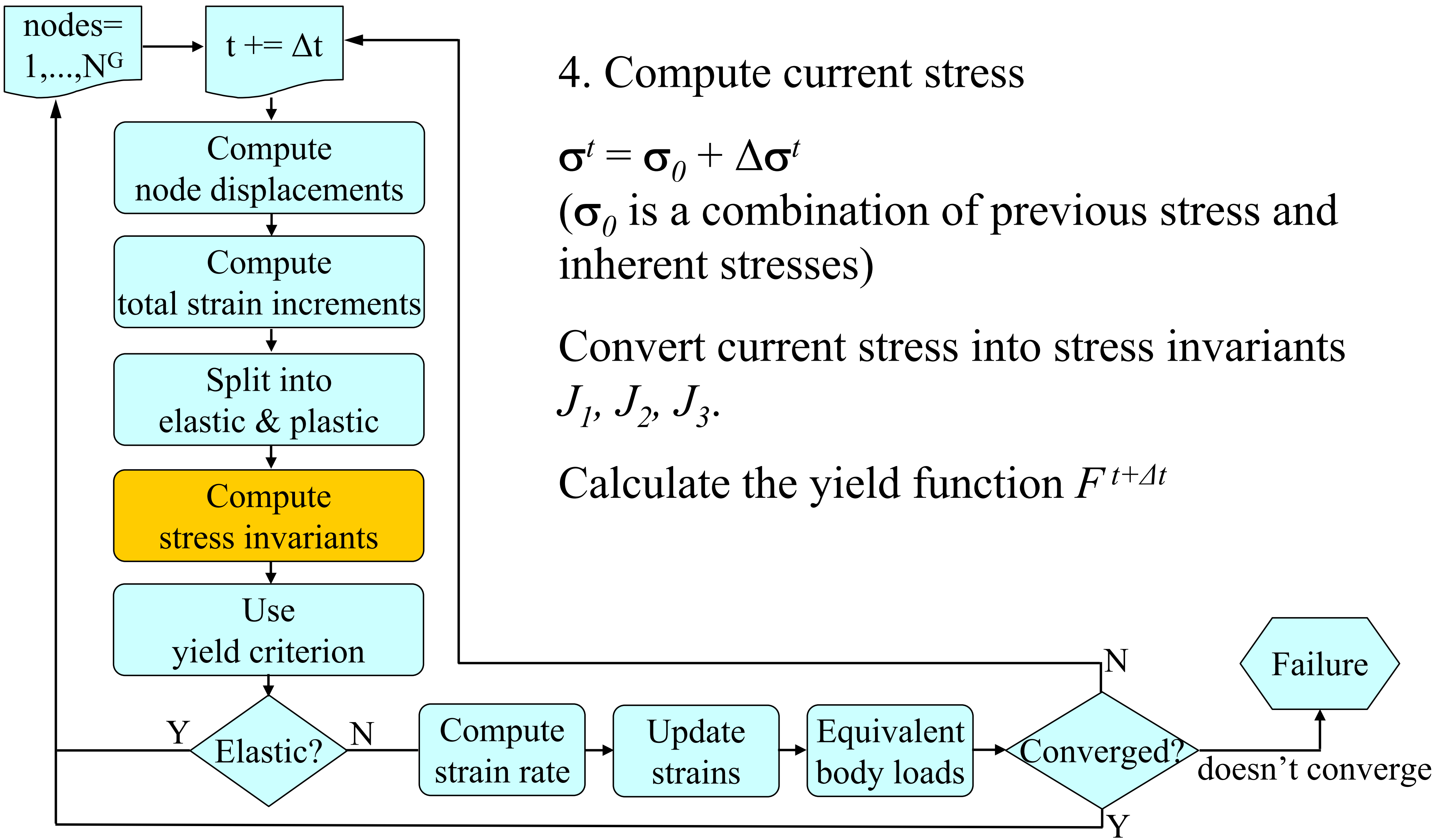


3. Estimate the elastic and visco-plastic parts of the strain

$$\Delta\boldsymbol{\sigma}^t = \mathbf{C}\{\Delta\boldsymbol{\varepsilon}^t - (\Delta\boldsymbol{\varepsilon}^p)^t\}$$

(At iteration 1 and in all unyielded FEM-nodes $(\Delta\boldsymbol{\varepsilon}^p)^t = 0$)

Visco-plasticity flow chart



4. Compute current stress

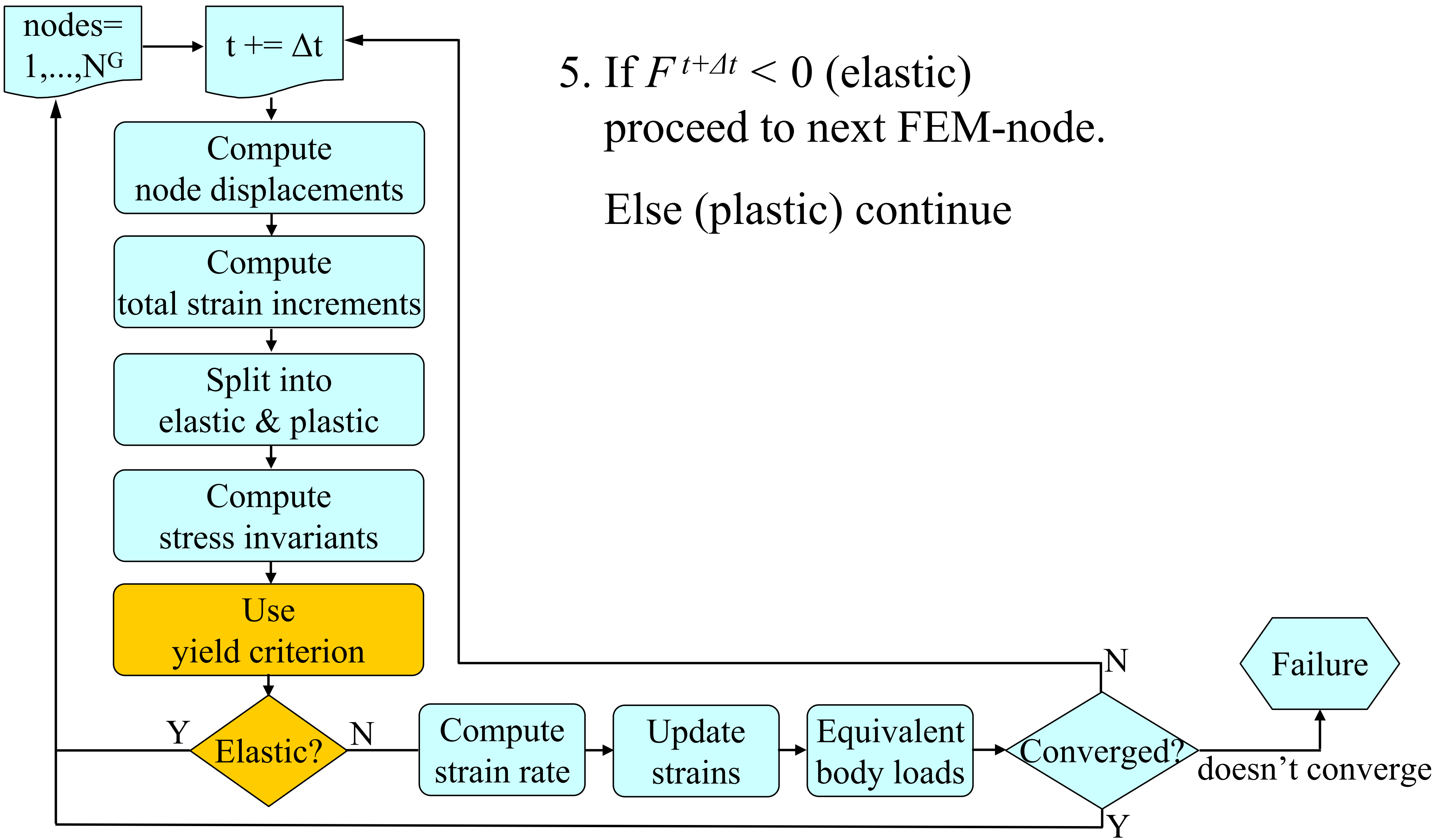
$\boldsymbol{\sigma}^t = \boldsymbol{\sigma}_0 + \Delta\boldsymbol{\sigma}^t$

($\boldsymbol{\sigma}_0$ is a combination of previous stress and inherent stresses)

Convert current stress into stress invariants J_1, J_2, J_3.

Calculate the yield function $F^{\,t+\Delta t}$

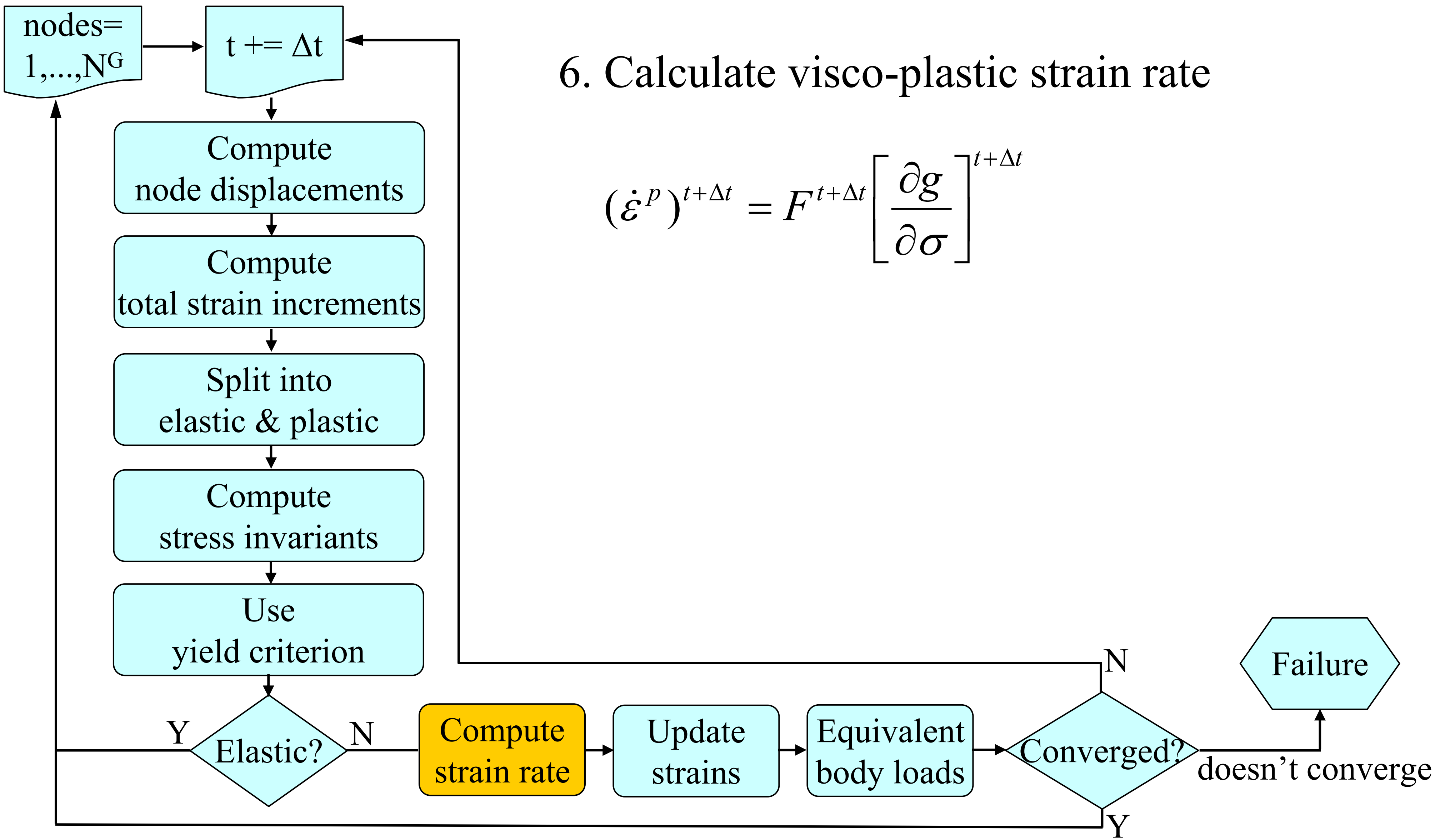
Visco-plasticity flow chart



5. If $F^{t+\Delta t} < 0$ (elastic) proceed to next FEM-node.

Else (plastic) continue

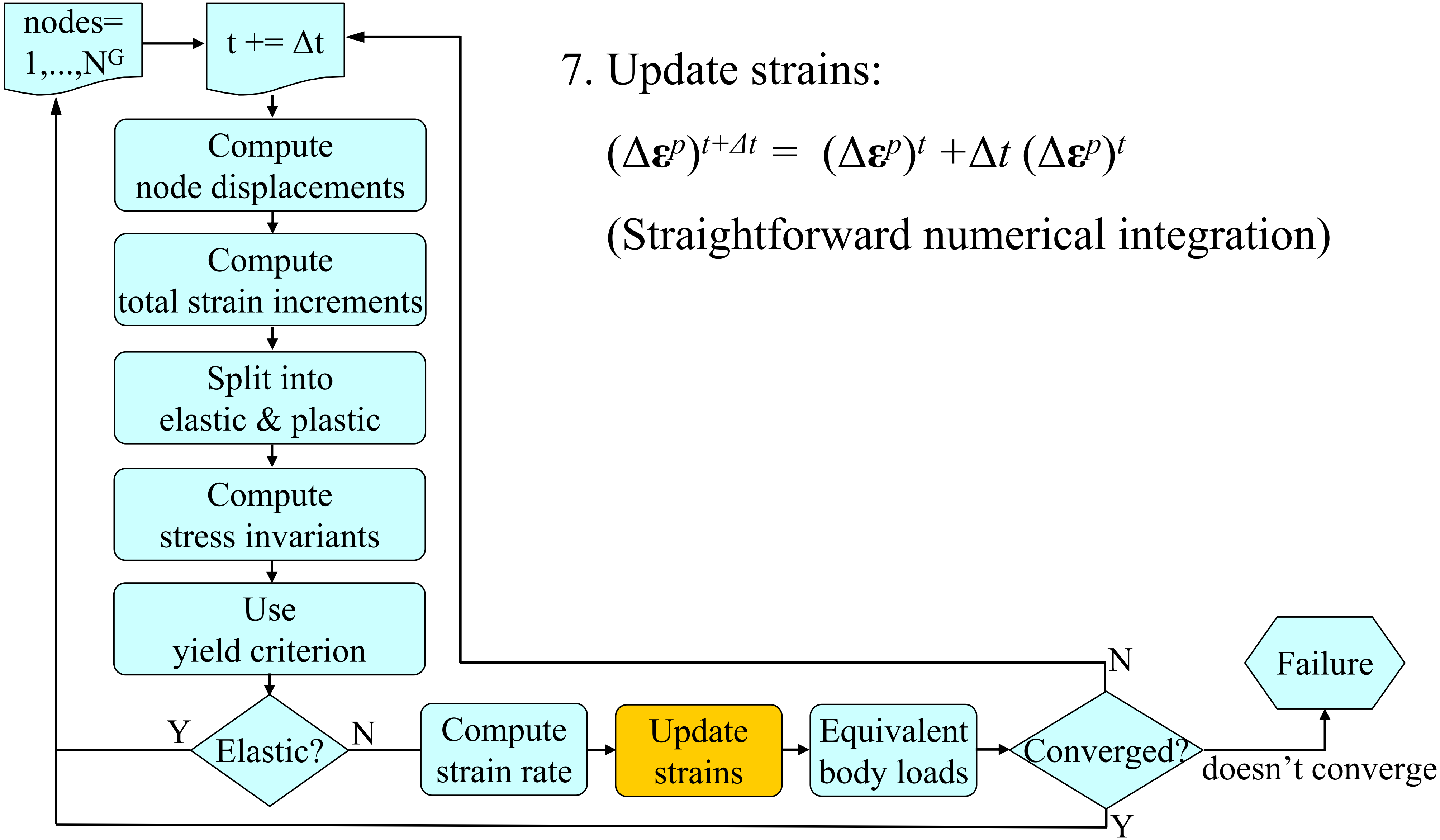
Visco-plasticity flow chart



6. Calculate visco-plastic strain rate

$$(\dot{\varepsilon}^{p})^{t+\Delta t} = F^{t+\Delta t}\left[\frac{\partial g}{\partial \sigma}\right]^{t+\Delta t}$$

Visco-plasticity flow chart

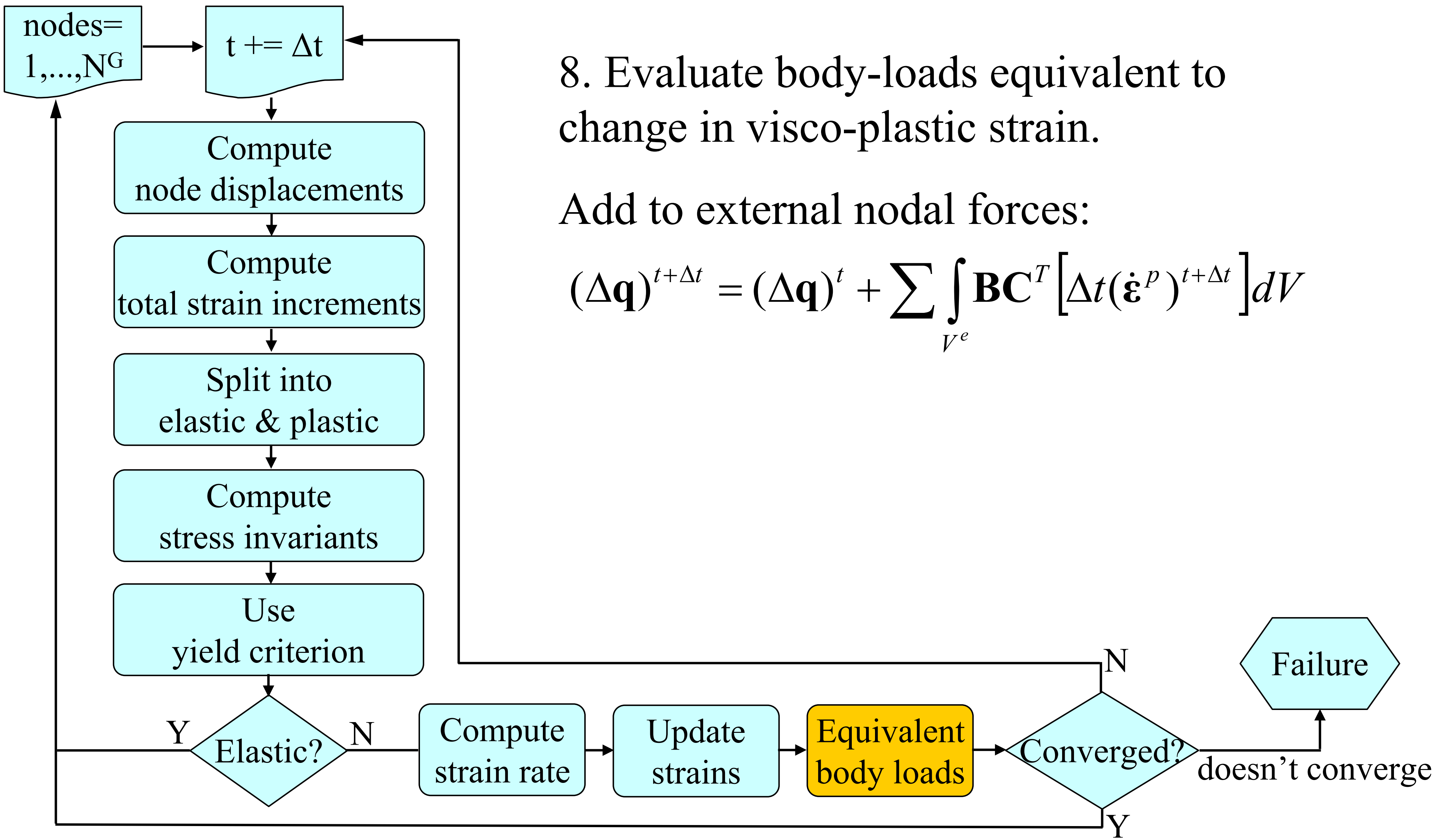


7. Update strains:

$$(\Delta\boldsymbol{\varepsilon}^p)^{t+\Delta t} = (\Delta\boldsymbol{\varepsilon}^p)^t + \Delta t\, (\Delta\dot{\boldsymbol{\varepsilon}}^p)^t$$

(Straightforward numerical integration)

Visco-plasticity flow chart

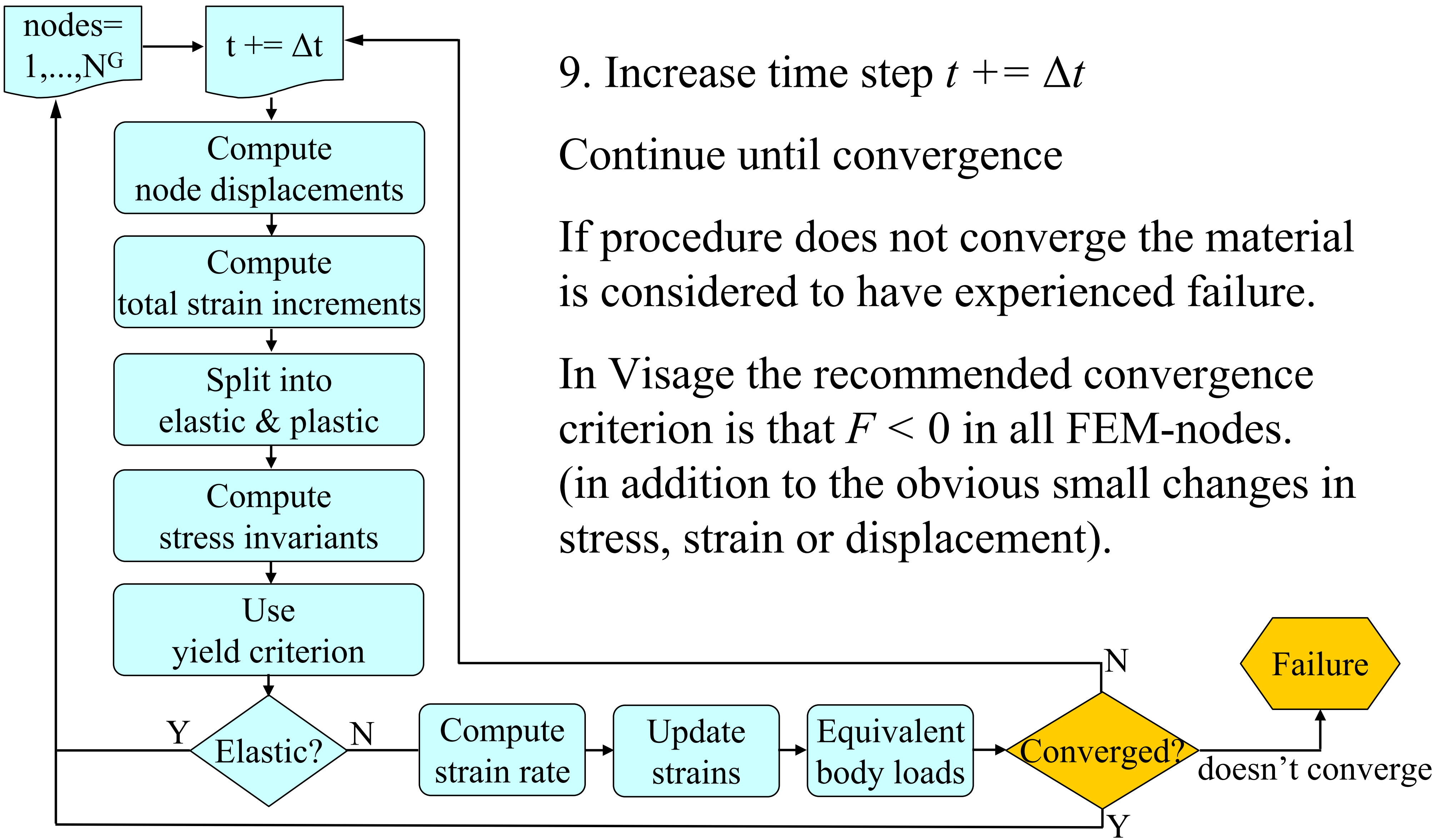


8. Evaluate body-loads equivalent to change in visco-plastic strain.

Add to external nodal forces:

$$(\Delta\mathbf{q})^{t+\Delta t} = (\Delta\mathbf{q})^{t} + \sum \int_{V^e} \mathbf{BC}^T \left[\Delta t (\dot{\boldsymbol{\varepsilon}}^p)^{t+\Delta t}\right] dV$$

Visco-plasticity flow chart



9. Increase time step t += Δt

Continue until convergence

If procedure does not converge the material is considered to have experienced failure.

In Visage the recommended convergence criterion is that $F < 0$ in all FEM-nodes. (in addition to the obvious small changes in stress, strain or displacement).