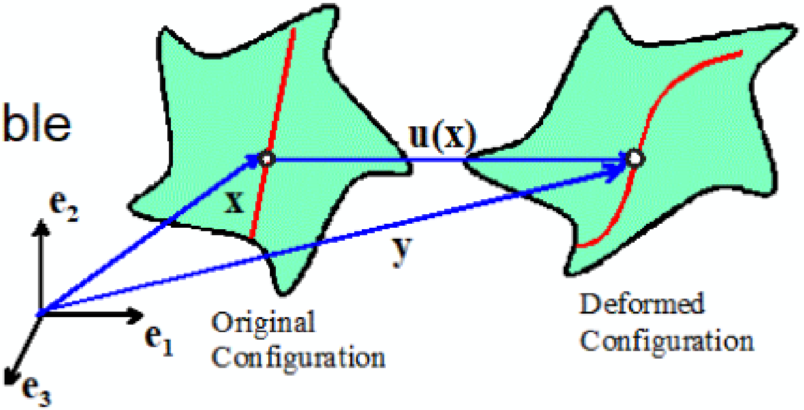
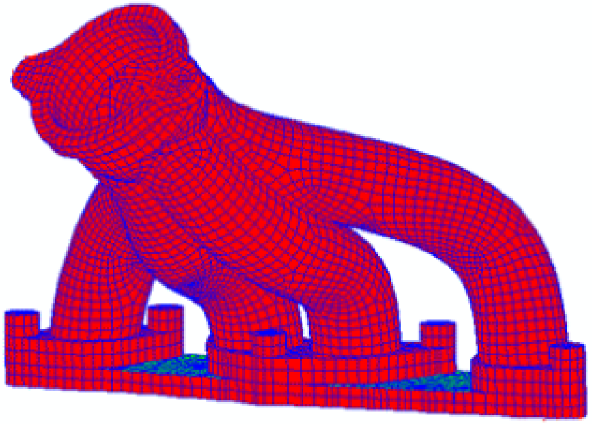
Review



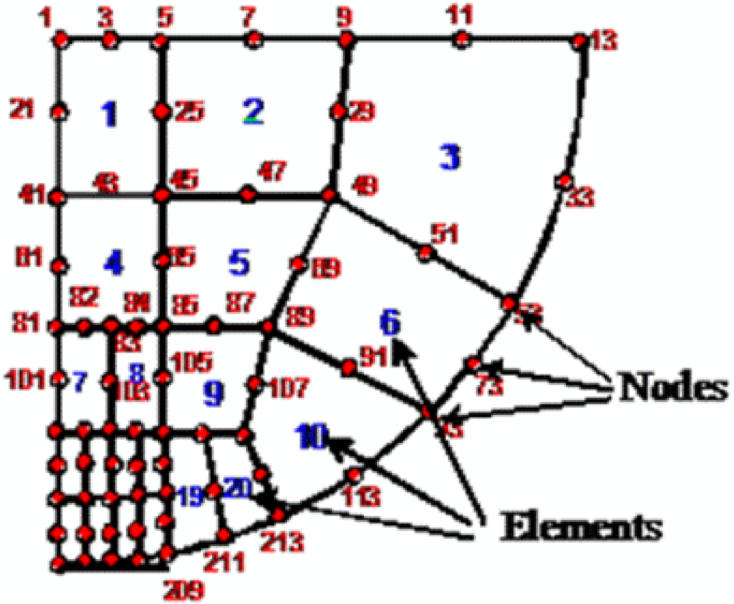
- Goal of FEA – compute displacements in deformable solid subjected to forces

- Steps in FEA
  - Define geometry – 3D or 2D
  - Interpolate displacements between discrete points
  - Material Properties
  - Governing equations
    - **F=0**
    - **F=ma**
  - Small-v-large deformations
  - Loading/boundary conditions; contact and interfaces
  - Solution method –
    - Static – linear or nonlinear
    - Dynamic – explicit or implicit



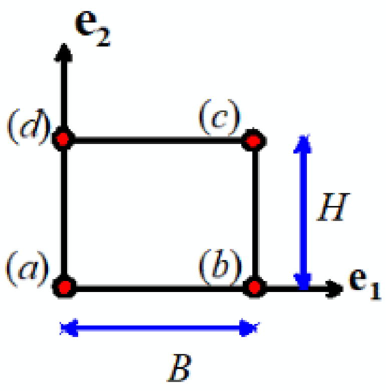
Review

- FE Mesh



- **Nodes** – discrete points, given initial coords. Nodal displacements are unknowns. Special elements may have rotational DOF at nodes

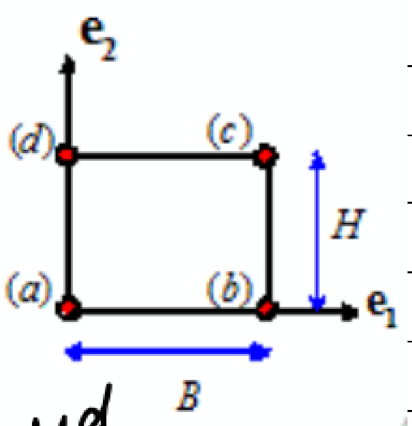
- **Elements** – subdivide solid into regions. Elements always have nodes at all vertices, and may have other nodes. Element properties:
  - List of nodes attached to element (connectivity)
  - Integration points (special points inside element where output variables are computed)
  - Interpolation functions (used to compute displacements within element)



Topics for todays class

- Element properties (continued)
- Boundary conditions
- Constraints
- Contact
- Solution procedures in FEA
  - Static analysis
  - Dynamic analysis
- Output from FEA

Element interpolation functions



Used to calculat u at arbitrary point inside element - eg for linear rectangle:

Define $\xi = x_1/B$ $\eta = x_2/H$

$$\underline{u} = (1-\xi)(1-\eta)\underline{u}^a + (1-\eta)\xi\underline{u}^b + \xi\eta\underline{u}^c + (1-\xi)\eta\underline{u}^d$$

ABAQUS also has quadratic elements

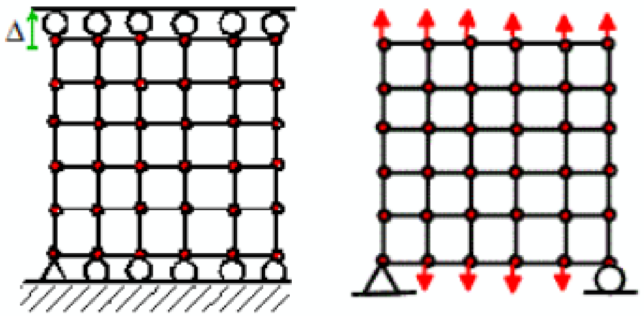
Choice of interpolation is important to ensure convergence

2.2 Material Models

ABAQUS has big list of common material models

- Can also implement custom models

2.3 Boundary Conditions / Loading



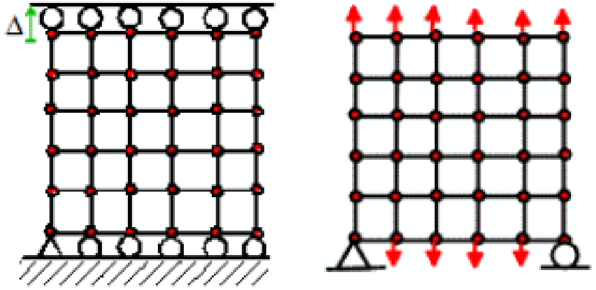
For solid elements you can:

(a) Prescribe $\underline{u}$ at nodes
(b) " force on node
(c) Pressure on element faces
(d) Body forces inside elements

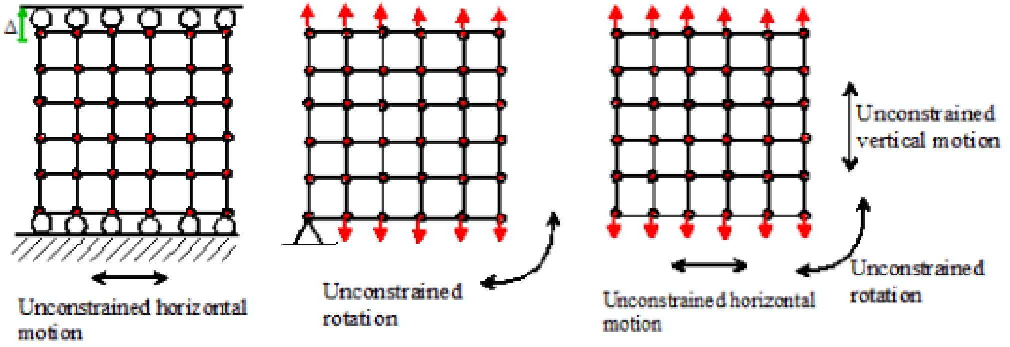
For beam / plate / shell, can prescribe rotations
can apply moments

In a static analysis you usually need to constrain rigid body motion

OK:

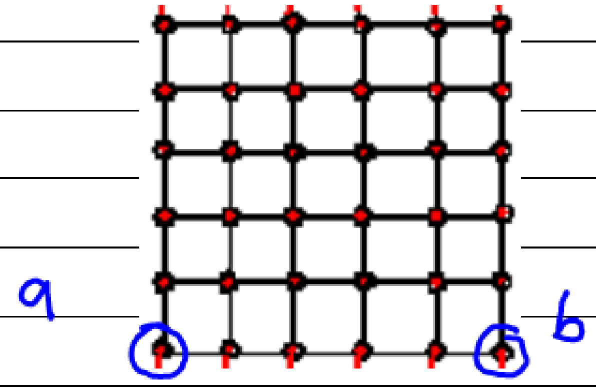


No:



2.4 Constraints

Equations relating displacements @ nodes can be enforced with constraints "MPC" in ABAQUS

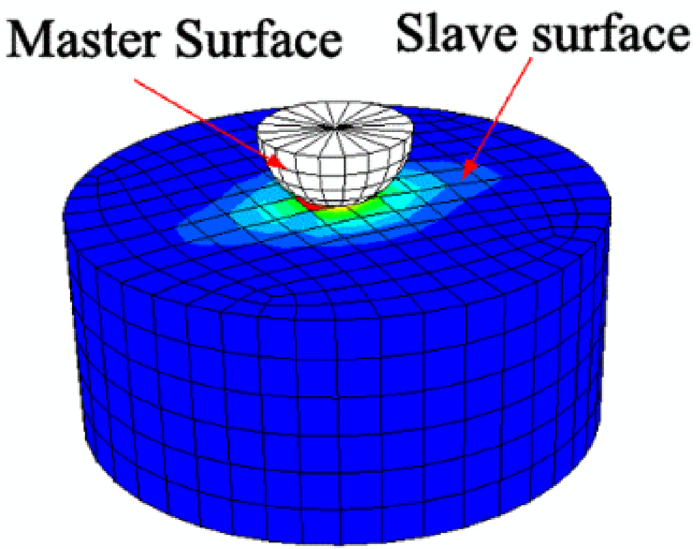


Can also connect beams or plates to solid elements

Periodic BC:
$u_i^a - u_i^b = \varepsilon_{ij}(x_j^a - x_j^b)$

2.5 Contact

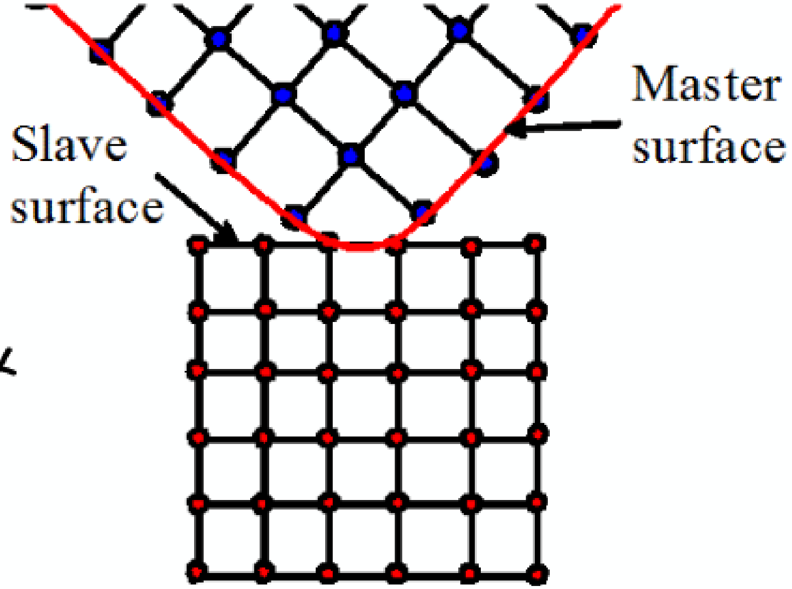
Enforce constraints where solids touch



2 options in ABAQUS :
"Node Based"
"Surface Based"

"Node Based" contact

Prevents nodes on "slave" surface from penetrating "master"



"Surface based" – enforces constraint in average sense on element faces – usually gives better contact pressure

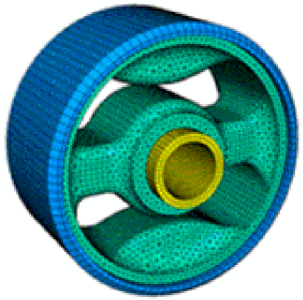
Also need to define "properties" of contact

Normal : "Hard" contact ⇒ no interpenetration
no tensile forces

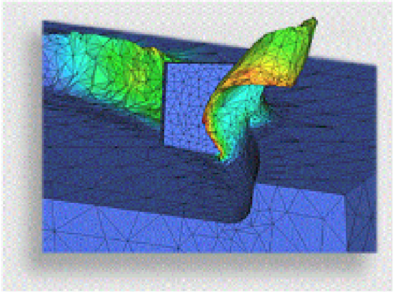
"Soft" ⇒ surfaces may interpenetrate
Need to define pressure-overclosure

Tangential : Choose friction law

2.6 Solution Procedures in FEA



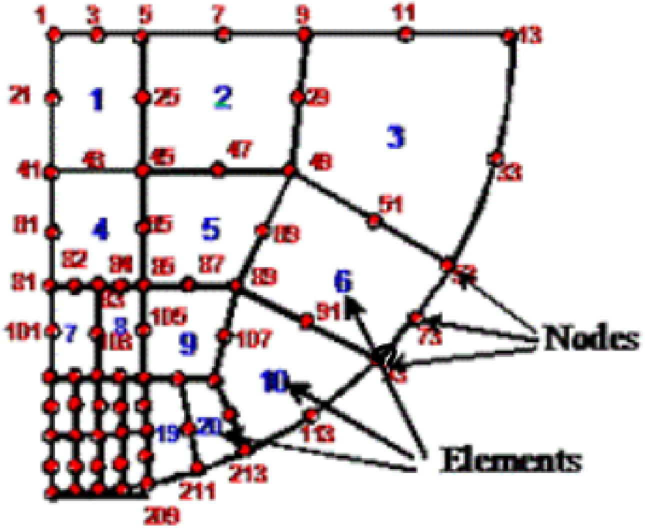
Small Def



Large Def

- * Small or large deformation
  NLGEOM
  Small def is approximate, but equation often linear ⇒ faster
- * Static or dynamic
- * "Linear perturbation" analysis - eg modal dynamics or buckling analysis

2.6.1 Static Solution procedure



For static analysis we need to solve

$$\underbrace{\underline{R}^{a}(\underline{u}^{b})}_{\text{Internal force at node a}} = \underbrace{\underline{F}^{*a}}_{\text{External force}} \qquad \underline{u}^{a} = \underset{\substack{\uparrow \\ \text{Prescribed Value}}}{\underline{u}^{a*}}$$

- System of nonlinear equations to solve for $\underline{u}^a$
- Solved using Newton-Raphson iteration

Newton-Raphson method

(a) Guess solution $\underline{u}^a$

(b) Corrected guess satisfies $\underline{R}^a(\underline{u}^b + d\underline{w}^b) = \underline{F}^{*a}$

(c) Linearize $\underbrace{\left.\frac{\partial \underline{R}^a}{\partial \underline{u}^b}\right|_{d\underline{w}=0}}_{\text{"Consistent tangent" - 3N×3N matrix}} d\underline{w}^b = \underline{F}^{*a} - \underline{R}^a(\underline{u}^b)$ ← ↑

$\underline{u}^a \to \underline{u}^a + d\underline{w}^a$ ↑

(d) Solve linear system for $d\underline{w}^b$

(e) Convergence check eg $|d\underline{w}^b| < tol$ → No ☹

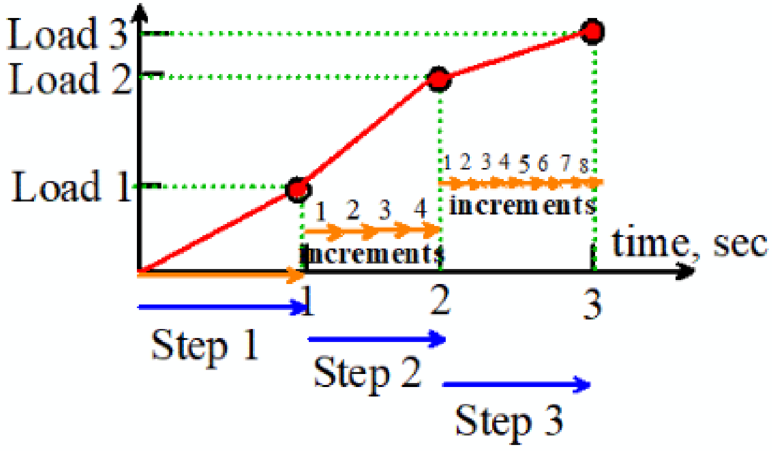
$|\underline{F}^{*a} - \underline{R}(\underline{u}^a)| < tol$

↓ Converged

Post-process

Newton iteration may not converge

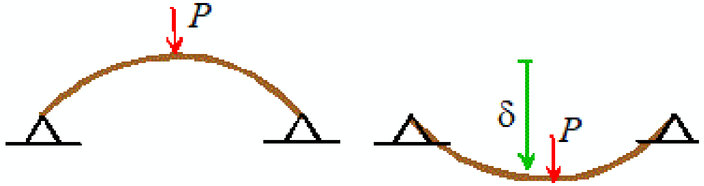
Helpful to apply load in small increments

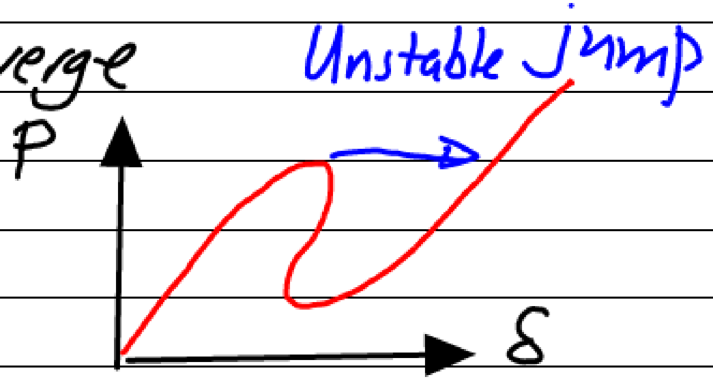


ABAQUS will automatically cut time increment if N-R fails

For some problems N-R will never converge

eg snap-thro buckling





Fixes: Implicit dynamics
Add artificial viscosity (Button in ABAQUS)
"Riks" method - follows unstable branch

2.5.2 Dynamics

FEA solves: $[m]\dfrac{d^2\underline{u}^a}{dt^2} = F^{*a} - R^a(\underline{u}^b)$

Need time stepping : Given $\underline{u}^a$ @ t
find $\underline{u}^a$ @ $t+\Delta t$

Explicit Dynamics

(a) Solve $\underline{a}^a = [m]^{-1}(\underline{F}^{*a} - R^a(\underline{u}^b))$

(b) $\underline{V}(t+\Delta t) = \underline{V}(t) + \underline{a}\,\Delta t$

(c) $\underline{u}(t+\Delta t) = \underline{u}(t) + \underline{V}\,\Delta t + \frac{1}{2}\underline{a}\,\Delta t^2$

- Explicit dynamics is conditionally stable
- Time step Δt < time for elastic wave to propagate thro smallest element in mesh for stability
- Speed up sometimes with "mass scaling"
  - multiplies density by scale factor to slow wave speed
    - Check total KE remains small